4 Understanding the Basics of the Injection Mold

4.1 Design Rules

There are many rules for designing molds. These rules and standard practices are based on logic, past experience, convenience, and economy. For designing, mold making, and molding, it is usually of advantage to follow the rules. But occasionally, it may work out better if a rule is ignored and an alternative way is selected. In this text, the most common rules are noted, but the designer will learn only from experience which way to go. The designer must ever be open to new ideas and methods, to new molding and mold materials that may affect these rules.

4.2 The Basic Mold

4.2.1 Mold Cavity Space

The mold cavity space is a shape inside the mold, "excavated" (by machining the mold material) in such a manner that when the molding material (in our case, the plastic) is forced into this space it will take on the shape of the cavity space and, therefore, the desired product (Fig. 4.1). The principle of a mold is almost as old as human civilization. Molds have been used to make tools, weapons, bells, statues, and household articles, by pouring liquid metals (iron, bronze) into sand forms. Such molds, which are still used today in foundries, can be used only once because the mold is destroyed to release the product after it has solidified. Today, we are looking for *permanent* molds that can be used over and

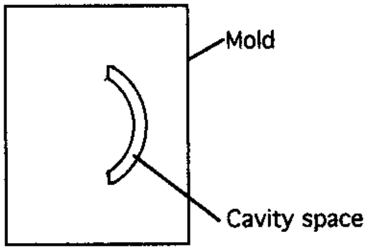

Figure 4.1 Illustration of basic mold, with one cavity space.

over. Now molds are made from strong, durable materials, such as steel, or from softer aluminum or metal alloys and even from certain plastics where a long mold life is not required because the planned production is small. In injection molding the (hot) plastic is injected into the cavity space with high pressure, so the mold must be strong enough to resist the injection pressure without deforming.

4.2.2 Number of Cavities

Many molds, particularly molds for larger products, are built for only 1 cavity space (a single-cavity mold), but many molds, especially large production molds, are built with 2 or more cavities (Fig. 4.2). The reason for this is purely economical. It takes only little more time to inject several cavities than to inject one. For example, a 4-cavity mold requires only (approximately) one-fourth of the machine time of a single-cavity mold. Conversely, the production increases in proportion to the number of cavities. A mold with more cavities is more expensive to build than a single-cavity mold, but (as in our example) not necessarily 4 times as much as a single-cavity mold. But it may also require a

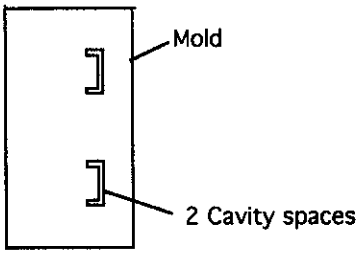

Figure 4.2 Illustration of basic mold with two cavity spaces.

larger machine with larger platen area and more clamping capacity, and because it will use (in this example) 4 times the amount of plastic, it may need a larger injection unit, so the machine hour cost will be higher than for a machine large enough for the smaller mold. Today, most multicavity molds are built with a preferred number of cavities: 2, 4, 6, 8, 12, 16, 24, 32, 48, 64, 96, 128. These numbers are selected because the cavities can be easily arranged in a rectangular pattern, which is easier for designing and dimensioning, for manufacturing, and for symmetry around the center of the machine, which is highly desirable to ensure equal clamping force for each cavity. A smaller number of cavities can also be laid out in a circular pattern, even with odd numbers of cavities, such as 3, 5, 7, 9. It is also possible to make cavity layouts for any number of cavities, provided such rules as symmetry of the projected areas around the machine centerline (as explained later) are observed.

4.2.3 Cavity Shape and Shrinkage

The shape of the cavity is essentially the “negative” of the shape of the desired product, with dimensional allowances added to allow for shrinking of the plastic. The fundamentals of shrinkage are discussed later.

The shape of the cavity is usually created with chip-removing machine tools, or with electric discharge machining (EDM), with chemical etching, or by any new method that may be available to remove metal or build it up, such as galvanic processes. It may also be created by casting (and then machining) certain metals (usually copper or zinc alloys) in plaster molds created from models of the product to be made, or by casting (and then machining) some suitable hard plastics (e.g., epoxy resins). The cavity shape can be either cut directly into the mold plates or formed by putting inserts into the plates.

4.3 Cavity and Core

By convention, the hollow (concave) portion of the cavity space is called the cavity. The matching, often raised (or convex) portion of the cavity space is called the core. Most plastic products are cup-shaped. This does not mean that they look like a cup, but they do have an inside and an outside. The outside of the product is formed by the cavity, the inside by the core. The alternative to the cup shape is the flat shape. In this case, there is no specific convex portion, and sometimes, the core looks like a mirror image of the cavity. Typical examples for this are plastic knives, game chips, or round disks such as records. While these items are simple in appearance, they often present serious molding problems for ejection of the product. Usually, the cavities are placed in the mold half that is mounted on the injection side, while the cores are placed in the moving half of the mold. The reason for this is that all injection molding machines provide an ejection mechanism on the moving platen and the products tend to shrink onto and cling to the core, from where they are then ejected. Most injection molding machines do not provide ejection mechanisms on the injection ("hot") side.

We have seen how the cavity spaces are inside the mold; now we consider the other basic elements of the mold.

4.4 The Parting Line

In illustrations Figs. 4.1 and 4.2 we showed the cavity space inside a mold. To be able to produce a mold (and to remove the molded pieces), we must have at least two separate mold halves, with the cavity in one side and the core in the other. The separation between these plates is called the parting line, and designated P/L. Actually, this is a parting area or plane, but, by convention, in this context it is referred to as a line. In a side view or cross section through the mold, this area is actually seen as a line (Fig. 4.3).

The parting line can have any shape, but for ease of mold manufacturing, it is preferable to have it in one plane. The parting line is always at the widest circumference of the product, to make ejection of the product from the mold possible. With some shapes it may be necessary to offset the P/L, or to have it at

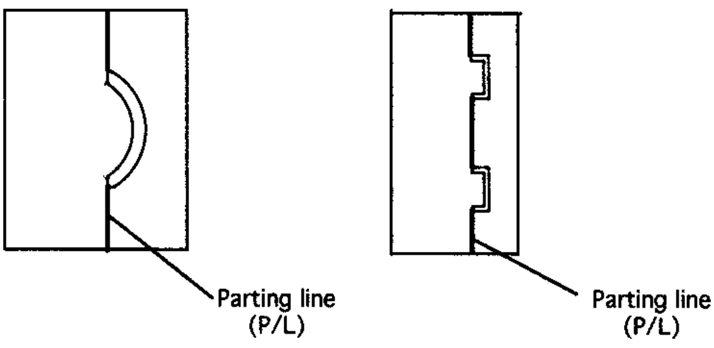

Figure 4.3 Illustration of schematic mold, showing the parting line.

an angle, but in any event it is best to have is so that it can be easily machined, and often ground, to ensure that it shuts off tightly when the mold is clamped during injection. If the parting line is poorly finished the plastic will escape, which shows up on the product as an unsightly sharp projection, or "flash," which must then be removed; otherwise, the product could be unusable. There is even a danger that the plastic could squirt out of the mold and do personal damage.

4.4.1 Split Molds and Side Cores

There are other parting (or split) lines than those that separate the cavity and core halves. These are the separating lines between two or more cavity sections if the cavity must separate (split or retract) to make it possible to eject the molded product as the mold opens for ejection.

Figure 4.4 shows simple "up and down" molds. The machine clamping force holds the mold closed at the P/L. (In (B) and (C), the parting line could be anywhere on the outside of the rim, between the two positions shown, but is preferred as in (B).) In (D) we must consider the injection pressure *p* (as shown with small arrows inside the cavity space), which will force the two cavity halves in the direction of the the large arrow *m*. This force also exists in the other examples, but is resisted by the strength of the solid cavity walls, which do slightly expand during injection and then return to their original shape once the injection cycle is completed. Since these side forces can be considerable (see Section 4.6), the mold plates (the "mold shoe") must be sufficiently solid to

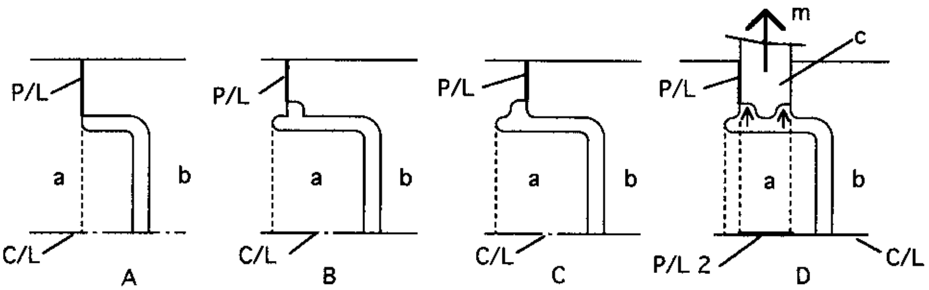

Figure 4.4 Schematic illustrations of location of parting lines (P/L) (only one half of mold shown): (a) core, (b) cavity. (A) Simplest case: P/L at right angles to axis of mold. (B and C) Product with rim but still simple. P/L can be either as in (B) or in (C). (D) Simple product but with rim and projection. Cavity is split, creating an additional P/L 2.

contain these forces and provide the necessary preload to prevent opening of the mold during injection. These side cores, or split portions of the cavities, can represent just small parts of the cavity, or even only small pins to create holes in the side of the products, but they could also be sections molding whole sides of a product, as, for example, with beverage crates or large pails.

4.5 Runners and Gates

In Fig. 4.3, we showed molds with cavity spaces and parting lines. Now, we must add provisions for bringing the plastic into these cavity spaces. This must be done with enough pressure so that the cavity spaces are filled completely before the plastic "freezes," that is, cools so much that the plastic cannot flow anymore. The flow passages are the sprue, from where the machine nozzle (see Fig. 3.1) contacts the mold, the runners, which distribute the plastic to the individual cavities, and the gates, which are (usually) small openings leading from the runner into the cavity space. We discuss the great variety of sprues, runners, and gates later. We illustrate here only two methods of so-called cold runners (see Fig. 4.5).

The left part of Fig. 4.5 shows the simplest case of a single-cavity mold, with the plastic injected directly from the sprue into the cavity space. This is a frequently used method, mostly with large products. It is inexpensive, but requires the clipping or machining of the relatively large (sprue) gate. The right drawing is of a typical (2-plate) cold runner system, with the plastic flowing through the sprue and the runner and entering the cavity space through relatively small gates, which break off easily after ejection. Instead of the 2 cavities as shown here, there can be any number of cavities supplied by the cold runners. These and other runner methods are explained later.

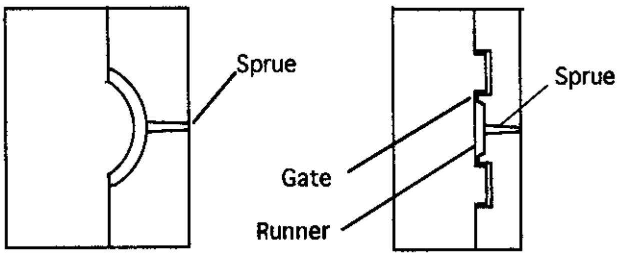

Figure 4.5 Illustration of schematic mold, showing cold sprue (left) and cold runner (right).

4.6 Projected Area and Injection Pressure

At this point we digress and consider injection pressure and how it affects mold design (see Fig. 4.6). As the plastic fills the cavity space under high pressure *p*, the pressure, in the direction of the mold (and machine) axis—in other words, in the direction of the motion of the clamp—will tend to open the cavity at the parting line. The separating force *F* created by the pressure *p* is equal to the product of the pressure *p* times the projected area *A*, which is the area of the largest projection of the product at the parting line. The arrow describing projected area in Fig. 4.6 really describes an area not a line, as delineated in this section view of the mold. The actual area can be seen (and measured) in a plan view of the mold cavity. From this it becomes clear that the clamping force, the force exerted on the mold by the molding machine, must be at least as great as the force *F* to keep the mold from opening (cracking open) during injection.

The difficulty is how to determine the value of the injection pressure *p*. We can easily calculate the injection pressure inside the machine nozzle, which is directly related to the size of the injection cylinder of the machine and the hydraulic (oil) pressure supplying the injection cylinder. The injection pressure at the machine nozzle, in general, is adjustable between any low values, to a high of about 140 MPa (20,000 psi), in most molding machines, and in some machines can be as high as 200 MPa (29,000 psi) or even higher. This pressure,

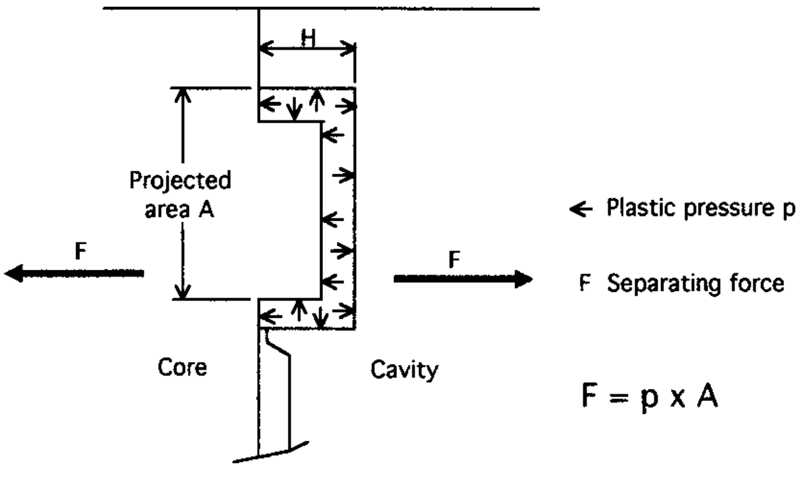

Figure 4.6 Portion of a schematic mold, showing a cavity filled with plastic under pressure acting in all directions.

however, is greatly reduced (by the pressure drop) by the time the plastic passes through the machine nozzle orifice, the runners, and the gates, and as it flows through the narrow passages of the cavity space. The flow also depends largely on the viscosity (defining the ease of flow) of the plastic, which depends on its chemistry and on its temperature (the higher the temperature, the lower the viscosity). This area is the subject of much research and experimentation, and computer programs are available to calculate the pressures and the flow inside the cavity space (see Appendix).

A good working assumption is a cavity pressure p of approximately 30–40 MPa (4000–5000 psi) for average product wall thicknesses of about 2–3 mm or more, and 40–50 MPa (5000–6000 psi) or even higher for thin-wall products. For example, a disk of 100 mm (10 cm) diameter, with a thickness of 2 mm, will generate an opening force of $(10^2 \times \pi \div 4)\,\mathrm{cm}^2 \times 30\ \mathrm{MPa} = 235\ \mathrm{kN}$ (approx. 26 US tons) per cavity.

4.6.1 Clamping Force

From the above example we see that a clamping force of at least 235 kN (26 US tons) per cavity should be used to ensure that the mold will not crack open. If the average wall of the product is thinner, or if the definition, that is, the accuracy and clarity of reproduction of details in the cavity wall, is important, then the pressure must be higher and a larger clamping force will be required.

4.6.2 Strength of the Mold

There are two other serious effects of the injection pressure p. First, as can be seen in Fig. 4.6, the pressure also acts in the direction at right angles to the axis of the mold. These forces, which are the product of the projection of the cavity in this direction times the pressure p, will tend to stretch and deflect the cavity walls outward. The greater the height H of the product, the greater will be this force and the stronger must be the walls surrounding the cavity.

Second, the clamping force is applied as soon as the mold closes. At this moment, the whole clamp force is resisted ("taken up") by the area of the land, which is the area surrounding the cavity that touches the core side. If this area is too small, the land will be crushed and damage the sealing-off surfaces of the parting line, eventually ruining the mold. Proper sizing of the land and correct materials and hardness (steel, etc.), or other measures to counteract the clamping forces are the solution to this problem. Also, the mold setup technician should be informed by a nameplate attached to the mold that the recommended maximum clamp force for the mold must not be exceeded during mold setup or during operation.

4.6.3 Why Are High Injection Pressures Needed?

High injection pressures are needed to ensure that the mold is completely filled during the injection cycle, with the desired clear surface definition. There are several problems to consider.

(1) The thinner the wall thickness of the product, the more difficult it is to push the plastic through the gap between cavity and core, thus requiring higher pressures. Since material (the plastic) usually accounts for 50–80% of the total cost of a molded product, it is highly desirable to reduce the weight (mass) of plastic injected to a bare minimum. This usually means reducing the wall thickness as far as possible without affecting the usefulness of the product. Over the years, many products have been redesigned just to reduce the plastic mass of a product. This is also why many modern injection molding machines provide higher injection pressures than older ones.

(2) The colder the injected plastic, the higher its viscosity, and the more difficult it becomes to fill the mold. The cost of the product depends directly on the cycle time required to mold a product. The higher the melt temperature of the plastic, the easier it will flow and fill the mold. However, higher melt temperatures also require increasing the cooling cycle time to bring the temperature of the injected plastic down to a level where the product can be safely ejected without distorting or otherwise damaging it. This means more power (for heating and cooling), longer cycles, and therefore higher costs. It is often better to inject at the lowest possible temperatures, even if more pressure is needed to fill the mold. Note that higher injection pressures will require greater clamping forces and a stronger, possibly larger, machine. Another solution to the problem might be to select a plastic that flows more easily. Such plastics, however, are usually more expensive and may not be as strong as desired.

(3) High injection forces are needed for good surface definition. Typically, this is important when molding articles such as compact discs, where the clarity

and precision of the surface definition is in direct relation to the quality of the sound reproduction of the recording.

4.7 Venting

As the plastic flows from the gate into the cavity space, the air trapped in it as the mold closed must be permitted to escape. Typically, the trapped air is being pushed ahead by the rapidly advancing plastic front, toward all points farthest away from the gate. The faster the plastic enters—which is usually desirable—the more the trapped air is compressed if it is not permitted to escape, or vented. This rapidly compressed air heats up to such an extent that the plastic in contact with the air will overheat and possibly be burnt. Even if the air is not hot enough to burn the plastic, it may prevent the filling of any small corners where air is trapped and cause incomplete filling of the cavity. Most cavity spaces can be vented successfully at the parting line, but often additional vents, especially in deep recesses or in ribs, are necessary.

Another venting problem arises when plastic fronts flowing from two or more directions collide and trap air between them. Unless vents are placed there the plastic will not "knit" and may even leave a hole in the wall of the product. This can be the case when more than one gate feeds one cavity space, or when the plastic flow splits in two after leaving the gate, due to the shape of the product or the location of the gate. Within the cavity space, plastic always flows along the path of least resistance, and if there are thinner areas, they will fill only after the thicker sections are full.

Venting is discussed more thoroughly in *ME*, Chapter 11.

4.8 Cooling

Cooling and productivity are closely tied. In injection molding, the plastic is heated in the molding machine to its processing (melt) temperature by adding energy in the form of heat, which is mostly generated by the rotation (work) of the extruder screw. After injection, the plastic must be cooled; in other words, the heat energy in the plastic must be removed by cooling, so that the molded piece becomes rigid enough for ejection. Cooling may proceed slowly, by just letting the heat dissipate into the mold and from there into the environment. This is not suitable for large production, but for very short runs "artificial" cooling of a mold is not always required. However, for a production mold, good cooling to remove the heat efficiently is very important.

4.8.1 Basics of Cooling

The physics and mathematics of cooling are quite complicated. Computer programs can determine the appropriate means of cooling a particular mold, after input of the geometry of the product and the mold, and based on assumed temperatures of melt and coolant, flow patterns and sizes of the cooling channels, and other variables, such as heat characteristics of the coolant and the mold materials. This means that a computer program can determine the best planned cooling layout for a mold only *after* the mold is designed. But the designer wants to know how to design the best cooling layout in the first place. There are several rules, based on experience, to help the designer.

- **Rule 1:** Only moving coolant is effective for removing heat. Stagnant coolant in ends of channels, or in any pocket, does nothing for cooling.
- **Rule 2:** All cavities (and cores) must be cooled with the same coolant flow (quantity of coolant per unit of time) at a temperature that is little different from cavity to cavity (or core to core). The coolant temperature will rise as it passes through each cavity (or core), but this is the very purpose of the coolant: to remove heat, which will raise its own temperature. As long as the temperature difference ΔT between the first and the last cavity in one group of cavities (or cores) is not too large—on the order of $\Delta T = 1-5\,°C$ (2–9 °F), depending on the job—the system is working properly. The smaller the difference, the more coolant will be required (which is more expensive in operation). In many molds there can be a good argument for compromise by having a greater ΔT and thereby using less coolant. In some cases, however, the lowest ΔT value may be necessary for quality requirements of the product. This may require special coolant capacity and pumps.
- **Rule 3:** The amount of heat removed depends on the quantity (volume) of coolant flowing through the channels in cavity (or core). The faster the coolant flows, the better it is, because (a) a greater volume will flow through the channels, and (b) there will be less temperature rise of the coolant from the first to the last cavity (or core).
- **Rule 4:** The coolant must flow in a turbulent flow pattern, rather than in laminar flow. Turbulence within the flow causes the coolant to swirl around as it flows, thereby continuously bringing fresh, cool liquid in contact with the hot metal walls of the cooling channels, and removing more heat. By contrast, laminar flow moves along the channel walls

relatively undisturbed, so that the outer layer of the coolant in touch with the metal will heat up, but the center of the coolant flow will remain cold, thus doing little cooling.

Turbulent flow is defined by the Reynolds number (Re), which is calculated as $\mathrm{Re} = (V \times D) \div v$, where V is the velocity of the coolant (m/s), D is the diameter of the channel (m), and v is the kinematic viscosity (m^2/s). $v = \mu \div \rho$, where μ is the absolute viscosity ($kg/m \cdot s$), and ρ is the density of the coolant (kg/m^3). A Reynolds number of more than 4000 ($\mathrm{Re} > 4000$) designates turbulent flow. The higher the number, the better the cooling efficiency. For good cooling, $10{,}000 < \mathrm{Re} < 20{,}000$ should be attempted. For water at 5 °C (41°F), $\rho = 999.5\ kg/m^3$, $\mu = 1.55 \times 10^3\ kg/m \cdot s$, and $v = 1.5508 \times 10^{-6}\ m^2/s$. (More values can be found in *ME*, in Table 25.2.)

Thus, where cooling is important—in cavities, cores, inserts, side cores, and so on—small-diameter channels and fast-flowing coolant are also important. Most cooling lines for cavities and cores are supplied from channels in the underlying or surrounding plates, and can be much larger, therefore having a much smaller Re number. But this is usually satisfactory because these plates do not need as much cooling as the stack parts, which come in contact with the hot plastic.

■ **Rule 5:** Serial or parallel flow? (See Fig. 4.7.) It does not matter whether the coolant follows a serial flow, that is, from cavity to cavity (or core to core) in sequence (Fig. 4.7a), or whether the flow is split so that the coolant flows in a parallel pattern (Fig. 4.7b), as long as each branch has the same flow. In many multicavity molds, the cooling channels are arranged so that they are partly in parallel and partly in series (Fig. 4.7c). Often, in the same mold, cavities are in one arrangement of series, parallel, or both, and cores, inserts, or side cores, are in another arrangement, whichever is more suitable for the layout. There is no rule for which way to go, as long as the flow rules are followed.

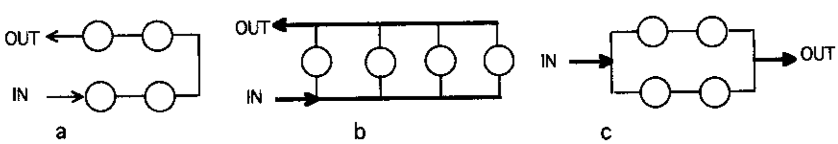

Figure 4.7 Schematic layout of (a) series cooling, (b) parallel cooling, and (c) series–parallel cooling.

■ **Rule 6:** The channel sizes (cross sections) must be calculated so that there is always more than enough flow capacity in a preceding section to feed equally all the channels in the following split, parallel sections. For example, if there are 4 parallel channels of $40\,mm^2$ cross-sectional area each, the (preceding) feeder must have at least $4 \times 40\ mm^2 = 160\ mm^2$ cross-sectional area. In some molds there are 4 or more points where the cross sections step down in the cooling system. It does not matter if the preceding section is greater than the calculated minimum value, but it must not be smaller, if the coolant is to flow equally through all subsequent channels. Coolant, like plastics, always takes the path of least resistance. For example, if the preceding cross section is $3x$, and each of 4 succeeding parallel cross sections are x, there will not be enough coolant, and one of the 4 channels will see little or no flow through it. Unfortunately, this is often missed in designs and the mold does not function properly.

- **Rule 7:** The difficult-to-cool areas in the mold must be considered first. These are, essentially, all delicate mold features, such as thin and slender core pins, blades, and sleeves. Slender signifies, in this context, that the ratio of length over the narrow bottom dimension or diameter of a pin or insert is more than 2 to 1. Remember that heat always flows from the higher toward the lower temperature; the flow decreases as the length of travel increases and as the cross-sectional area through which the heat travels gets smaller. Difficult-to-cool areas limit the mold cooling capability and seriously affect the molding cycle. There is no sense in providing good cooling for the easy-to-cool areas of the mold if there are poorly cooled areas elsewhere in it. Selecting materials such as beryllium–copper alloys may help to remove the heat faster, or special cooling methods may be used, such as blowing (cold) air at the thin sections while the mold is open. But first the designer must try to find a way of getting coolant (not necessarily water) into the thin sections, or at least get the best cooling into the mold parts supporting these thin projections.
- **Rule 8:** Study the product to locate heavy sections of the plastic. They are always a problem, even where it is easy to provide good cooling, because of potential shrink and sink marks. Heavy sections are particularly bad if they are toward the end of the plastics flow where there is less pressure to ensure good filling. The mold designer should discuss this problem with the product designer. There may be the possibility of a minor alteration of the product design to avoid heavy sections so that not only is plastic saved but also cooling time is reduced. For example, the heavy, solid handle of a coffee mug could be redesigned

by coring it from both sides. This could add to the mold cost, but would greatly reduce the cycle time. The question is whether the customer wants to sacrifice design features for productivity. (See also *Understanding Product Design for Injection Molding*.)

4.8.2 Plate Cooling

An often overlooked fact is that mold cooling is not only for cooling the plastic, but also for cooling the various mold plates that are close to areas heated by the plastic, such as the hot runner systems discussed later or, in special cases, such as injection blow molding, where the mold cores are heated to keep the plastic hot, for blowing immediately after injection. As is explained in Section 4.10, all materials expand when heated. In many molds, certain plates are essential for the alignment system because they carry the leader pins and bushings or other alignment members. If the mold plates are at different temperatures, they will expand differently from their original, cold state, and cause misalignment between the alignment elements. For example, assume that the distance of two leader pins in a mold is $L = 400$ mm and that a temperature difference of $\Delta T = 10\,°C$ (18 °F) exists between the two plates carrying the pins and bushings. With an approximate heat expansion for steel of 0.000011 mm/mm/°C, L will increase by ΔL. $\Delta L = L \times \Delta T \times 0.000011 = 400 \times 10 \times 0.000011 =$ 0.044 mm (0.00173 inch). Considering that the standard diametrical clearance between leader pins and bushings is only 0.025 mm (0.001 inch), the example shows the pins will bend at every cycle, or bind in the bushings. This points to the importance of ensuring in the design that both mold halves should be kept as close as possible to the same temperature. (Compression molding, usually employed for thermosetting materials, requires heating of the mold, regardless of productivity. In this process, the plastic must be heated to set (or harden); the product leaves the mold hotter than the raw material used to fill the mold.)

More about cooling later. See also *ME*, Chapter 13.

4.9 Ejection

After the plastic in the cavity spaces has cooled sufficiently and is rigid enough and ready for removal, the mold halves move apart, allowing sufficient space between the mold halves for removal of the product. As with cooling, the complexity of any provision for ejection from the mold is a question of the desired productivity. Some products don't need any provision within the mold for ejection. For example, a quick blast from an air jet applied manually by an operator and directed at the parting line can lift a (simple) product off the core or out of the cavity, but this would not be practical in most molds, and is rarely used for real production. Usually, the products are ejected by one of the following methods:

(1) Pin (and sleeve)
(2) Stripper plate or stripper ring
(3) Air alone
(4) Air assist
(5) Combination of any of the above (1), (2), (3), and (4)
(6) Unscrewing, in case of screw caps, etc.
(7) Combination of any of the above, combined with robots

The most common and oldest methods are

- Pin (and sleeve) as shown in Fig. 4.8
- Stripper plate or stripper ring, as shown in Fig. 4.9

  These two systems can be used in most molds and for most plastics. The problem with both these systems is that there are heavy moving parts involved, and the upkeep of such molds is high.

- Air ejection alone can be used for flat products (Fig. 4.10, *left*), but for deep cup-shaped products (*right*) it is restricted to only certain plastics and shapes. The main advantage is that it has no, or almost no, moving

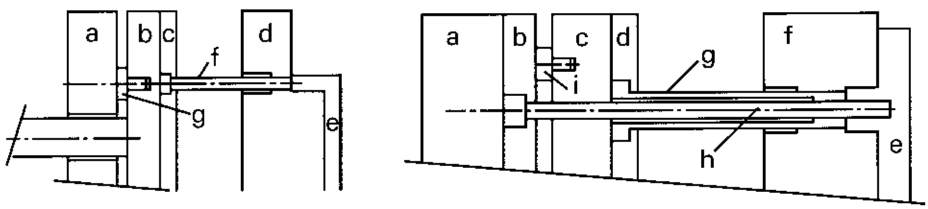

Figure 4.8 (*Left*) Section through ejector pin mold: (a) backing plate, (b) ejector plate, (c) ejector retainer plate, (d) core plate, (e) molded product, (f) ejector pin, (g) stop pin. (*Right*) Section through sleeve ejector mold: (a) backing plate, (b) core pin retainer plate, (c) ejector plate, (d) sleeve retainer plate, (e) molded product, (f) core plate, (g) sleeve ejector, (h) core pin, (i) stop pin.

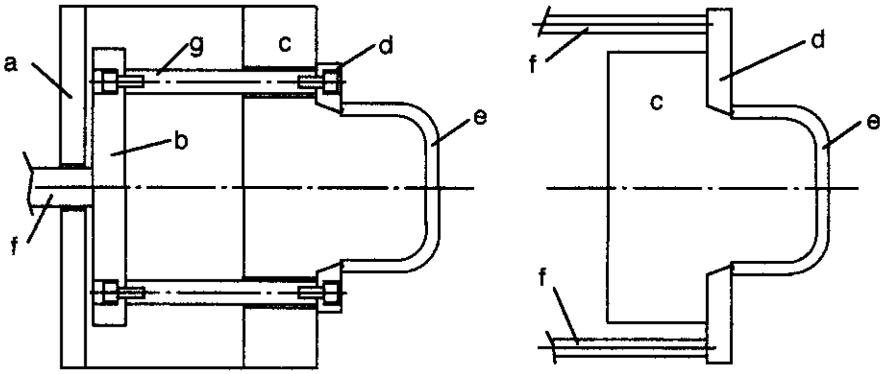

Figure 4.9 (*Left*) Section through stripper ring mold: (a) mounting plate, (b) ejector plate, (c) core plate, (d) stripper ring, (e) molded product, (f) machine ejector, (g) connecting sleeve. (*Right*) Section through stripper plate mold: (c) core and mounting plate, (d) stripper plate, (e) molded product, (f) machine ejectors.

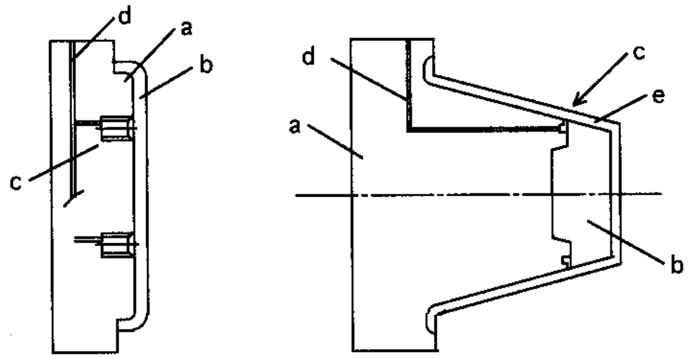

Figure 4.10 Air ejection alone. (*Left*) (a) core and mounting plate, (b) molded product, (c) air valves, (d) pressure air supply. (*Right*) (a) core and mounting plate, (b) core tip, (c) circular air gap, (d) pressure air supply, (e) molded product.

parts. Air ejection alone is often used in very high production molds; the same applies to (7), by combining any of the above ejection methods with integrated robots.

Note that for best productivity, to reduce cycle time, the products should be ejected as early as possible. Certain ejection methods permit earlier ejection; others depend on the plastic to be stiffer. For example, stripping permits hotter (softer) products to be ejected without damage to them, whereas unscrewing requires the pieces to be more rigid.

4.9.1 Automatic Molding

Earlier molds were all designed to require operators (often lowly paid and unskilled) to sit or stand at the molding machine. After every cycle they opened the safety gate to remove the products from the molding area, reclosed the gate and initiated the next molding cycle. They also were, in some cases, supposed to visually inspect the products at this time and even make adjustments to the machine if they thought it necessary. Because the molds were often not properly finished, by today's standards, or had unreliable injection and ejection systems, the operator was also often required to reach into the molding area to pry loose a stuck, possibly defective product, and from time to time had to lubricate the molding surfaces with mold release agents. All this was not only labor intensive, adding greatly to the cost of production, but was also very unsafe and the cause of many serious injuries. Since much of this operation also depended on the acquired skill of the operator—some workers are faster, some slower—and on the time of the day or night, or even on the day of the week, the overall molding cycle time could vary considerably, resulting in quality differences of the product because of different residence times of the melt in the machine; many rejects resulted. There was also the problem of absenteeism of the personnel, which often played havoc with production planning. Much effort was therefore spent on eliminating operators from the actual molding process.

Fully automatic (FA) molding depends essentially on two factors:

(1) *Reliable injection*. The molding machine must be repetitive from cycle to cycle in every aspect, but especially in the dosing (the amount of plastic injected) and the melt temperature.
(2) *Reliable ejection*. This is 100% the responsibility of the mold designer. Every mold (with very rare exceptions) can be designed so that there is no chance of the product hanging up and not ejecting. The key to good ejection is that the product always stays on the side from which it will be ejected, usually, but not necessarily, from the core side of the mold. The designer must select the appropriate method of ejection and make sure that there is enough ejection stroke to clear the products from the cores. This is frequently overlooked and can also be caused by improper setup of the mold. Many areas must be considered in the design; some are discussed later.

The designer must keep in mind Murphy's law, which says that if it can happen, it will.

See also *ME*, Chapter 12.

4.10 Shrinkage

One of the most misunderstood areas of mold design is shrinkage. Every material (metals, plastics, gases, liquids) expands as its temperature increases (heat expansion) and returns to its original volume if cooled down to the original temperature. The problem with all plastics is the characteristic of compressibility. All solid materials compress under load, but most not as much as plastics. When pressure is applied to plastics (or to hydraulic oil, but not to water), plastics will compress significantly (i.e., reduce in volume) in proportion to the amount of pressure applied. This may be (within the range of molding operations) as high as 2% of the original volume. Thus, we now have two conditions that work against each other: heat expansion and compressibility. As the plastic is injected, it is both hot and therefore expanded, but also under significant pressure, which reduces its volume. This makes it very difficult to arrive at a true shrinkage factor, because the actual change in volume depends on the type of plastic, the melt temperature, the injection pressure required to fill the cavity space, and the temperature at which it will be ejected from the mold.

For practical purposes, and for many products and molds, the shrinkage factors supplied by materials suppliers can be used. However, these figures indicate only a range within which to choose, usually between 0 and 5%. In some cases, where the volume or size of a product is important, this is not accurate enough. With crystalline plastics, such as polyethylene (PE), polypropylene (PP), and polyamide (nylon), the shrinkage factor is much higher than with amorphous plastics, such as polystyrene (PS) and polycarbonate (PC). Plastics filled with inert substances, such as glass or carbon fibers or talcum, have a much lower shrinkage than that for the same but unfilled material. Shrinkage figures should be obtained from materials suppliers, for guiding purposes.

4.10.1 Variable Shrinkage

The designer must understand that the areas within the cavity spaces close to the gate see higher pressures, so the shrinkage there will be less and will require a smaller shrinkage factor. Conversely, near the end of the flow through the narrow cavity space, the pressure in the plastic is much lower than near the gate, and a higher shrinkage factor will apply. In some applications, more than two shrinkage factors may have to be selected within one cavity. It is also important to establish at what temperature the product will be ejected. If it is ejected while still hot, it will shrink more outside of the cavity space as it cools to room temperature. If ejected later, when it is cooler, it will shrink less, as measured in comparison with the steel sizes of the cavity and core.

This is sometimes, but uneconomically, used to arrive at the proper size of a product such as a container or lid. If a molded product is too small because not enough shrinkage value was added to the product dimensions when specifying the mold steel dimensions, the proper product size can be achieved by ejecting it later, when it is cooler, but this means loss in productivity. With high production, the proper procedure is to resize the steel dimensions.

See also *ME*, Chapter 8.

4.11 Alignment

Various methods are used to align cavity and core plates. The method selected depends on the shape of the product, the accuracy (or tightness of tolerances) of the product, and even on the expected mold life. Several choices are available:

(1) No provision for alignment within the mold
(2) Leader pins and bushings
(3) Taper lock between each cavity and core
(4) Taper lock between a group of cavities and cores
(5) Wedge locks
(6) Taper pins
(7) Combination of (2) with (3), (4), (5), or (6)

4.11.1 No Provision for Alignment

In the case of a flat product, without any cavity (depression) in one mold half, and the cavity entirely in the other mold half, for example, in a mold for a floor mat, there is no need for alignment, even if there is some engraving on the flat surface of the mold, because the most the dimensions can vary is by the amount of play between the machine tie bars and the tie bar bushings.

4.11.2 Leader Pins and Bushings

This common method of alignment between mold halves is shown in Fig. 4.11. In cup-shaped products with heavy walls, there is really no need for alignment within the mold, because the clearances between tie bars and their bushings are usually much less than the tolerances of the product wall thickness. The main reason to have leader pins in these cases is to protect the projecting cores from physical damage, when handling the mold.

The protection of the cores by use of leader pins applies also to all other mold alignment methods. Wherever leader pins are used, they should be placed at the same mold side as the cores and be longer than the longest projection of the cores to protect them from damage (see dimension s, in Fig. 4.11). There are exceptions to this rule, for example, in some 3-plate molds.

What is often missed is that for most applications leader pins and bushings are a very accurate method of alignment. Consider dimension t in Fig. 4.11, and let's assume a wall thickness $t = 1.50$ mm (0.060 inch), with a tolerance of ± 0.05 mm (0.002 inch), or 1.50 ± 0.05 mm. With standard commercial hardware, the leader pin is usually nominal size minus 0.025 mm (-0.001 inch), and the bushing is nominal size plus 0.025 mm ($+0.001$ inch). Therefore, with one set of pins and bushings, the maximum clearance, in the highly unlikely worst case, between one set of leader pins and bushings could be 0.05 mm (0.002 inch) on the diameter, so the centers would be misaligned only half that amount. By having at least 2, but usually 4 sets, the total clearance between the pins in all the bushings would be even less. In the worst case, the

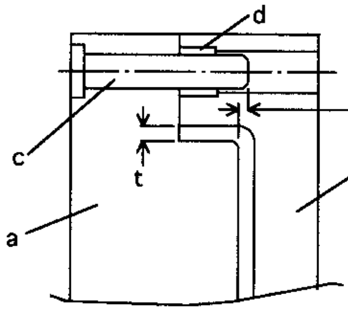

Figure 4.11 Typical mold with leader pin and bushing alignment: (a) core plate, (b) cavity plate, (c) leader pin, (d) leader pin bushing, (s) safety distance of pin above core, (t) wall thickness of plastic product at parting line.

possible play and misalignment would be well within the tolerance limits specified in this example, and therefore acceptable.

It can be easily seen that this holds true as long as the product has not much smaller wall thicknesses, as is often the case with thin-wall containers, with wall thicknesses in the order of 0.4 mm (0.015 inch) or even less. In those special but frequent cases, other methods of alignment must be used such as taper fits. We also must not forget the influence of heat expansion of the mold plates, which will affect the alignment accuracy.

4.11.3 Taper Lock Between Each Cavity and Core

Figure 4.12 shows 3 possible configurations of taper or wedge locks. On the left, the tapers in both male and female members match perfectly. Because of manufacturing tolerances, this is impossible to achieve except, perhaps, by individual fitting of parts, and even then it is difficult. To be able to produce any mold part without need for fitting (center), they must be closely toleranced and accurately machined. To solve the problem of providing proper alignment, the matching parts are dimensioned such that the male member is slightly larger than the female member, and the female member will be slightly expanded from the moment the mold halves touch, until the mold is fully clamped. The amount that the pieces stay apart before final clamping (d) is called preload in Fig. 4.12. This amount *d* is very, very small, and depends on the length of the taper and on its angle. It must be greater than zero. On the right, the female member is larger than the male member. This taper lock is useless because the tapers don't touch (f); no force is generated to pull the mold halves into alignment.

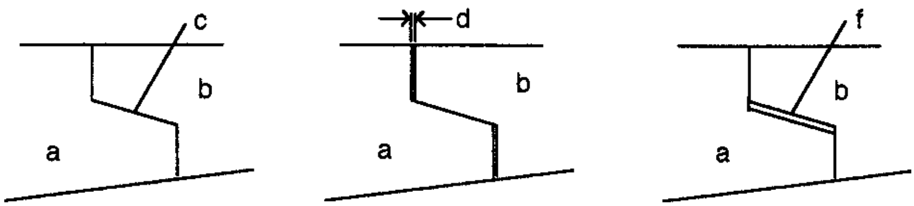

Figure 4.12 Taper (or wedge) lock: (a) male member, (b) female member, (c) taper. (*Left*) Ideal condition. (*Center*) Correct application. *d* is called preload. (*Right*) Useless taper.

In practice, it can be easily seen on a mold if the tapers work: If the tapers (or wedges) are shiny all around, they work; if they are rusty, or just dirty, they don't work, and the mold probably depends on the tie bars and tie bar bushings for alignment, or on the mold leader pins and bushings. It is surprising how many molds are in this category. Many times the designer (or the mold maker) thought that by providing tapers, the mold will be more accurately aligned. In most of these cases, the taper fit was wasted money. Note that working tapers are subject to severe wear and must be made from suitable, hardened steels, and even so will have to be replaced or repaired from time to time. Any size taper is acceptable, between 5 and 20 °. (Common tapers are 7, 10, and 15 °.) Too small a taper may cause locking and separation difficulty because of friction in the tapers; too large a taper requires too much force to close. Obviously, to move the tapers for the preload distance *d*, until they seat properly, means that the matching, female taper will have to be spread. This requires considerable force. When considering the clamp force of the machine, this must be considered and the forces calculated, especially with multicavity molds in which every stack is aligned with taper locks. If too much force is required for closing the mold, there may not be enough clamp force left for holding the mold closed during injection.

4.11.4 Taper Locks and Wedges

Taper locks are conical (usually round) matching mold parts, and the taper of the cone is designed to provide the alignment between two mold parts (cavity–core, core–stripper ring, etc.). This method is very accurate and relatively inexpensive, but has two inherent disadvantages:

(1) The alignment of the various components depends on the accuracy of machining and once the assembly is finished, there is no possibility of adjusting the alignment.

(2) Once the tapers wear, which is unavoidable due to the very nature of this design, which must touch and rub, they are difficult to repair and reuse without changing other mold parts as well. The easiest way is often to replace the worn elements.

Wedges are pairs of hardened, flat bars, with one side tapered. Four sets of wedges are always required per alignment, either for each cavity, or for the whole mold. The advantage is that wedges can be shimmed or ground on opposite pairs to adjust for wear or for inaccurate manufacturing, or easily replaced if shimming is not practical. The disadvantage of wedges is that they require more space on the mold surface, so the mold size will be larger than when using taper locks.

4.11.5 Taper Pins

Taper pins (and bushings) are sometimes used for the final alignment of cavity and core in addition to leader pins, where it is believed that the accuracy of leader pins is insufficient. They act similarly to taper locks and are available as standard mold hardware. It is questionable whether they do any better job than the other methods of alignments explained here; and they are subject to the same problems as taper locks, regarding wear and accuracy of machining the mold and/or core plates.

4.11.6 Too Many Alignment Features

Another problem is frequently encountered in poorly designed molds. Typically, cavities and cores can be aligned by either leader pins and bushings, or taper (or wedge) locks. Where high accuracy in alignment is required, taper (or wedge) locks are the preferred choice. However, they do not assure that the mold halves will stay together when handling the mold; there is always the danger that the cores and cavities could be damaged if the mold halves should separate and bang together once the taper engagement is lost. It is therefore necessary to equip the mold with leader pins (but not necessarily with leader pin bushings), in addition to the taper locks. Since the tapers will determine the final alignment, the leader pins must fit only loosely in their corresponding openings (or leader pin bushings) without actually contributing to the final alignment of cavities and cores. Quite often, even for large molds, only two such pins need to be provided, usually located at the top of the mold on the core side.

Similarly, some multicavity molds are built with small leader pins (usually only two) and bushings for each set of cavity and core and are mounted on the stack plates; they ensure the final alignment of each stack. In addition, two or four large leader pins are used to align the complete mold halves, but these pins also must be “loose” in their bushings, to prevent “fighting” between the two separate sets of alignments. An exception to this rule of loose pins is when a more expensive but superior method is used: the cores are mounted such that they can move slightly (float) on their backing plates; as the mold closes, the final alignment (tapers or pins) will move each core into position relative to its cavity. In this case, the leader pins mounted in the mold shoe (on the core side) will have their regular, standard clearances.