runner system takes care of the gate location. This method overcomes the objection of the first method where the off-center mold puts unequal stress on the tie bars. A molding machine has been developed which has the injection carriage move laterally on the bed as well as axially. The platen is correspondingly cored so that the injection cylinder can be brought to meet an off-center sprue.

2. Insert molding modifications. The need for insert molding, particularly molding vinyl electrical cord sets, led to the development of a shuttle table machine. This is a vertical machine with one core plate attached to the movable platen. While one set of cavities (A) is molding, the operator removes the previously molded parts and inserts new inserts into the other identical cavity section (B). When the cycle is done A slides out and B slides under the force and the cycle is repeated.

3. Two color molding modifications. The need for two color molding, such as the numerals in typewriter keys, led to the development of machines with more than one injection cylinder. Many ingenious mold designs are used to transfer the very first molding and use it as a matrix for the second molding.

4. Rotary and multistation machines. A rotary type machine with a turret system was developed for rapid production of dissimilar parts with relatively low mold costs. As the turret moves from station to station, the material is injected, cured, and ejected. While one cavity is being molded the other parts are cooling. The individual mold stations can be equipped with different cavities of different weights, thicknesses, and dimensions. This system is easier to control than a conventional multicavity mold. There is also more time available for unscrewing threaded parts. In the event a cavity is damaged, the machine can skip that part. Turret machines are considerably more complicated mechanically than the standard machines (75).
CHAPTER 2

Molds

The injection mold is the mechanism into which the hot plasticized material is injected and maintained under pressure while it cools into a commercially acceptable shape. When the plastic material has sufficiently solidified, the machine opens, separating the two halves of the mold. The plastic pieces are ejected (Figure 2-1). The quality of the part and its cost of manufacture are strongly influenced by mold design, construction, and excellence of workmanship.

As machine capacity increases, the molds become larger and more expensive. It is not uncommon, for example, for a mold to produce a garbage can and cover that costs over $25,000.00. Molds for 12-oz machines usually range from $3000 to $9000 (1). This is only a small part of the investment. The original idea, market testing, samples, prototypes, advertising, selling, and commitment for the initial order are many times the cost of the tool.

The two most critical steps in the production of a plastic part are the piece part design and the mold design. A failure of the first naturally results in an unacceptable part. Mold design failure does not necessarily result in piece part failure, although well it may. It does cause low productivity, high mold maintenance, and the probability of reduced part quality. An improperly functioning mold will take excessive supervisory time in the plant and tool maintenance department. This can cause neglect of other operations, compounding the normal problems of running a molding plant.

It is for this reason that the molder, the moldmaker, and on occasion, the purchaser, should give maximum attention to the design. Unfortunately this is not always done.

The last opportunity to change the part easily is when the mold is designed. In many instances the molder or purchaser will consult the moldmaker prior to ordering the mold. If not, suggestions by the moldmaker for changing the part to simplify the mold are to be given full consideration. The moldmakers are a versatile, ingenious, knowledgeable, and highly experienced group and have contributed much to the success of the industry. Moldmaking establishments
vary from small shops with the owner and two or more employees, to plants with several hundred employees and a full range of production equipment. There is no correlation between the size of the tool shop and the quality of its molds.

**Limits of Responsibility**

Let us consider the relationship between the molder, moldmaker, and user. The limits of responsibility and allocation of costs should be determined initially. The following are the current trade practices which have worked quite successfully. If the end user is the molder, he will be fully aware of his costs and the relationship with the particular moldmaker. If the moldmaker has a machine available for testing, he will submit molded parts for approval. The molder will then be reasonably assured of a mold that he can put into his machine and start producing immediately without any significant additional costs. This will be particularly true if he is able to watch the final molding test. If the molder tests the mold himself, he must include the costs of material, machine time, changes that might be made in the molder's machine shop, and the cost of transporting the mold back and forth to the mold maker. These costs can be substantial. The molder will evaluate them based on his estimate of the mold maker's ability, his experience with him, and the complexity of the mold.

If there is a third party, the end user, he can order the mold in two ways. He can order the mold through the molder, who will assume full responsibility for
delivering a workable mold and the acceptable molded part. In this instance the molder will include the costs of testing, either in the mold price that he quotes to the end user or in his piece part price.

If the end user buys the mold directly from the mold maker, he assumes the responsibility of delivering a working mold in the condition required to produce the part at the quoted price. He should assume full responsibility for the testing and other costs needed to make the mold operate.

Mold Design Procedure

The first step in designing a mold is to have an accurate fully dimensioned drawing that notes tapers and where they start, tolerances, shrinkage specifications, surface finish specifications, material in which it is to be molded, part identification, and any other pertinent information. It is desirable to have a model of the part to be molded. If the drawing or the part is complicated, this is a necessity. Mold design is significantly more reliable, and molding problems are more readily anticipated. Most models need not be made exactly to the sizes on the drawing. They should always be full scale and may be made of plastic, metal, plaster, clay, wood, cardboard, paper, or any other material.

There should be representatives of four plant functions during the mold designing stage—engineering, molding, mold maintenance, and quality control. While they need not meet together all the time, they should approve the final mold design. If any one of them feels there might be problems, these problems should be noted with the suggested remedies. Obviously the mold maker must agree to the design. At this time the metal for the cavity and the core will be selected. The cavity and core are those parts of the mold which, when held together by the closing of the mold, will provide the air space or cavity into which the molten plastic is injected. The following will now be decided: the number of cavities (2); the parting line of the piece (where the faces of the cavities and cores touch); the type and location of the gates (the entry point of the hot plastic into the cavity); the runner system (which brings the hot plastic from the plasticizing chamber to the gate); the method of ejection (which removes the molded plastic parts from the mold); the location of the ejecting devices; the location and size of the temperature control channels; and the type and location of the venting system (which removes the air that is displaced by the incoming plastic).

Any additional information required by the molder or moldmaker is discussed and drawings of the mold are submitted. These must be reviewed very carefully. A list of 73 items to take into account is found on p. 169 which should be checked against each mold drawing.

Studying a mold design and drawing is a time consuming and often ignored task. Actually, few activities are more productive for the molder and
moldmaker. It is much easier to change a drawing than a completed mold. A mold takes several months to complete so there is great urgency to produce molded parts immediately after the mold is delivered. Because of this, and the difficulty, cost, and time to make substantial mold changes, the minimum amount of work is done "just to get the mold running." Too often the mold remains in this condition as a monument to poor or careless mold design.

One cannot overemphasize the importance of getting a mold to function correctly. If this is not done, the foreman will be occupied with the problem for an excessive amount of time, preventing him from caring for the other machines; unacceptable parts will be passed by the operator and burden quality control and a poorer part will be produced. There is no more expensive error than bad tooling.

The state of the art has not advanced to the point where there is a complete guarantee that all parts can be molded, molded in the material selected, or molded at a cost consonant with the quoted price. The time to minimize these possibilities is when the mold is designed (3, 4).

In-House Moldmaking

Injection molders may have their own tool making facilities. This gives them control of the design, quality, and delivery of the tools. It permits them to expedite really urgent projects.

There are a number of disadvantages to in-house moldmaking. An outside source may be selected which has particular equipment or experience best suited for that particular job. He is less susceptible to pressures to put aside someone else's mold and concentrate on that of a good customer. He has a smaller capital investment and does not have the problem of managing and staffing a mold shop. Many molders compromise by having a small shop for some of their requirements and contracting out for the balance.

THE MOLD BASE

The steel parts which contain the cavities and cores are called a mold base, mold frame, mold set, die base, die set, or shoe. Figure 2-2 shows an exploded view of the parts of a standard mold base which are described below.

Seating Ring and Sprue Bushing

The locating or seating ring centers the sprue bushing on the stationary or injection platen of the machine directly in line with the nozzle of the injection cylinder. The sprue has an opening in the center of a concave spherical surface, whose counterpart is an equivalent convex surface on the nozzle of the injection
heating cylinder. The opening of the sprue bushing must always be larger than the opening in the nozzle, so when the plastic hardens it will not form an obstruction larger than the sprue opening and cause the sprue to stick. The sprue has a generous taper to facilitate removal of the plastic. The standard radii for sprue bushings and nozzles are either 1/2 or 3/4 in.

**Clamping Plates and Leader Pins**

The injection (top) clamping or another backup plate supports the cavity or A plate. Usually a pocket is cut through the A plate to hold the cavity. The cavities are attached to the backup or clamping plate or preferably in milled pockets in
the $A$ plate. The $A$ plate has four leader pins which are guided into four corresponding bushings in the $B$ plate. The $B$ plate has the cores attached to it or in pockets cut through it. Sometimes the location of the pins and bushings are reversed. The leader pins and bushings align the $A$ and $B$ plates and hence the cavities and cores. One pin is offset so the mold cannot be put together in the wrong position. The leader pins should be long enough so that if part of the mold slips during molding the leader pins will hit the $A$ or $B$ plates first, rather than a cavity or a core. When the leader pins or bushings wear, they can cause the $A$ and $B$ plates to shift several thousands of an inch with corresponding misalignment of the cavities and cores. If a container with an 0.018 wall was being molded and the leader pins and bushings wore and misaligned the cores and cavities by 0.004, then one wall of the container would be 0.014 and the other 0.022. This would cause severe filling, and ejection problems, and provide insufficient strength for the intended use. Leader pin wear problems are compounded by wear on the tie bars and bushings of the molding machine.

**Wedge Locking Device**

In critical applications a wedge locking device on the cavities and cores or the $A$ and $B$ plates is required. This will eliminate the effect of leader pin and bushing wear during molding. It will not cure ejection problems or the scratching of the part when the mold opens. Replacement of the pins or bushings will be required. If the machine was fully clamped at all times the angle of the wedge lock would not be important. Even with machines of adequate clamping capacity there will be times when the machine will open and the core and cavity separate during molding. Assuming a 0.006-in. opening, a 20° taper would permit the core to shift 0.002, a 45° 0.006, and a 60° taper 0.010. Other reasons for core shift are discussed later.

A stripper ring should not be used to align the cavity and core. The compression of the stripper ring when used for alignment may permanently deform it and will lead to excessive wear and possible galling on the core.

**Back-Up Plates**

The back-up or supporting plate under the $B$ plate serves the same function as the one behind the $A$ plate.

The B support plate rests on the ejector housing. Some molds have the ejector housing consisting of a back plate and two $C$ bar spacers. Now these parts are fabricated as one unit. Holes are drilled in the back of the ejector housing to permit the ejector bars of the machine to contact the ejector plate of the mold before the machine stops opening. In mechanical systems the ejector plate is stopped while the balance of the mold moves further back. This establishes a motion between the ejector plate and the $B$ plate on which are the molded
plastic parts, the common expression notwithstanding, "the ejector plate moves forward." In machines with hydraulic knockouts, the K.O. plate does move forward.

**Ejector Pins and Plates**

Ejector pins are made either from H-11 (a high chrome vanadium steel) or a nitriding steel. They are hardened to a surface hardness of 70 to 80 Rc to a depth of 0.004 to 0.007. The inside core is tough. The heads are forged for maximum strength. They are honed to a super finish. The K.O. pins come in fractional and letter size diameters. Each size can be had 0.005 oversize. These are used when K.O. holes are worn and flash occurs around the pins.

The ejector pins are countersunk into the ejector plate. The ejector retaining plate holds them in position when the two plates are screwed together. The ejector plate also contains four push-back pins. These pins are contacted by the A plate when the mold closes and pushes the knockout plate to its back position, so that the knockout pins and/or the cavities are not damaged.

**Knock-Out (K.O.) Mechanisms**

The proper way to open a mold is to use a bar to drive the K.O. plate forward. The push back pins will separate the mold. This prevents cocking the plates and possible damage to the leader pins, and is much superior to driving a wedge between the A and B plates.

In the center of the plate there is a sprue puller (pin) attached to the knockout plate, directly opposite the sprue. If a Z shaped undercut is machined on the sprue puller, plastic will mold around it. When the knockout plate moves forward, the pin is raised above the mold surface so that the runner can slide off the undercut. Other ways of pulling the sprue are to machine the top of the hole in which the sprue puller moves with a reverse taper (about 5° per side) or to machine a ring undercut (0.010 to 0.020 radius × 1/16 in. wide).

Sometimes the knockout arrangement of the machine falls outside of the mold base. Then the knockout plate is extended further than the rest of the mold or steel wings are bolted or welded to its sides. See "Ejection Systems" for additional data on ejection (p. 128).

**Support Pillars**

Most molds require an additional support in the area between the parallels of the ejector housing. These support pillars go through the ejection plate and have to be located without interfering with the ejection system.

The D-M-E Corporation, which pioneered standard mold bases and parts suggest the following calculations for support pillars:
\[ W = \frac{8ZS}{M} \quad Z = \frac{B^2L}{6} \quad A = \frac{W}{P} \]

where \( W \) = permissible load (pounds on support plate)
\( Z \) = section modulus (in.\(^3\))
\( A \) = permissible cavity area (in.\(^2\))
\( S \) = permissible working stress of steel (12,000 psi)
\( M \) = distance between supports (in.) (based on row of support pillars on 3 in. centers.)
\( B \) = thickness of support plate (in.)
\( L \) = length of support plate (in.)
\( P \) = effective polymer pressure on support plate (10,000 psi)

Combining these equations:

\[ A = \frac{1.6B^2L}{M} \] (2-1)

For a 15-in. plate, 1.875 in. thick, and 8½ in. between supports, the maximum projected molding area is 10 in.\(^2\). Obviously for a mold plate of this size, the projected area is too low. If a row of support pillars were added down the middle, reducing \( M \) to 4½ in., then each side could contain double the area and the plate four times or 40 in.\(^2\). If two rows were added then nine times the area is available or 90 in.\(^2\). One should be careful to have enough support pillar area so that the clamp force of the machine will not hob the pillars into the backup plate.

The obvious result of insufficient support is flashing. As the plate flexes other things may occur. The K.O. pins and pushback pins may bind. Cavities and cores may become angular in relation to each other causing ejection problems, parts scoring, and possible cracking. If the machine clamping tonnage is sufficient, the first place to check in case of trouble is the support pillar location, calculating the allowable projected molding area.

**Types of Bases Available**

A number of companies manufacture standard mold bases and parts. Because of their high volume, they have equipment which usually makes their mold bases less expensive and superior to those manufactured by the moldmaker. In addition, replacement parts are standard and readily available to the molder at a low cost.

There are cavities and cores of such size or shape where a mold base is best built around them. There are several types of steel available for the mold base. It is strongly urged that the best quality be used for any mold which requires high quality parts or long production runs.

Many mold bases are made from a medium carbon, silicon-killed forging
quality steel AISI-C type 1030 with a hardness range of 165-185 Brinell. A better grade steel, AISI type 4140 which is slightly more expensive and a little more difficult to machine is recommended. It comes normalized and drawn to relieve internal stresses and preheat treated to 252-302 Brinell. This steel can be hardened to Rc-45. Mold sets are also available in prehardened steel into which cavities and cores can be cut.

A rigid mold is essential and is the result of mold design, correct material selection of the mold frame, sufficient supporting pillars, and proper core, cavity, and gating. If the frame does not remain rigid the best cavities and cores will not stay in line. (A general discussion of molds is found in Ref. 75, 76, and 77.)

DESCRIPTION OF BASIC MOLD TYPES

The injection mold is identified descriptively by a combination of some of the following terms. They will be described in the text following.

Mold Parting lines
- Regular
- Irregular
- Two plate mold
- Three plate mold

Runner system
- Hot runner
- Insulated runner

Material
- Steel
- Stainless steel
- Prehardened steel
- Hardened steel
- Beryllium copper
- Chrome plated
- Aluminum
- Epoxy

Number of cavities

Methods of manufacture
- Machined
- Hobbed
- Cast
- Pressure cast
- Electroplated
- EDM (spark erosion)
**Gating**

<table>
<thead>
<tr>
<th>Edge</th>
<th>Diaphragm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Restricted (Pin Pointed)</td>
<td>Tab</td>
</tr>
<tr>
<td>Submarine</td>
<td>Flash</td>
</tr>
<tr>
<td>Sprue</td>
<td>Fan</td>
</tr>
<tr>
<td>Ring</td>
<td>Multiple</td>
</tr>
</tbody>
</table>

**Ejection**

<table>
<thead>
<tr>
<th>Stripper ring</th>
<th>Removable insert</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stripper plate</td>
<td>Hydraulic core pull</td>
</tr>
<tr>
<td>Unscrewing</td>
<td>Pneumatic core pull</td>
</tr>
<tr>
<td>Cam</td>
<td>Knock out pins</td>
</tr>
</tbody>
</table>

**PARTING LINE**

When a mold closes, the core and cavity meet, producing an air space into which the plastic is injected. If one were inside of this air space and looking toward the outside, this mating junction would appear as a line. It so appears on the molded piece and is called the parting line. A piece might have several parting lines if it has cam or side actions. The expression “parting line” is usually restricted to that line which is related to the primary opening of the mold. The mold separates at the parting line in a two plate mold and at the parting line, and runner plate in a three plate mold.

The selection of the parting line is largely influenced by the shape of the piece, method of fabrication, tapers, tolerances, method of ejection, type of mold, aesthetic considerations, post molding operations, inserts, venting, wall thickness, the number of cavities, and the location and type of gating.

Figure 2-3a shows a simple parting line on a flat plane. It is simple to make. The surfaces are ground flat. Figure 2-3b shows an irregular parting line. In this instance the shape consists of flat plane and arcs of circles which can be mechanically machined. These shapes are more difficult to make and maintain. Parting lines on figures or animals, for example, are completely irregular and usually require hand fitting.

Figure 2-4 shows a slab of plastic 3/16 in. thick, 1/2 in. wide, and 1 in. long. The shape of the molded piece will depend on the parting line chosen. The tapers have been exaggerated in the drawing. On a molded piece the eye readily picks up very small tapers. If the parting line is selected as in A, the two bottom edges would be sharp while the top could be sharp or radiused. This might not be acceptable for a handle as the sharp edge could be uncomfortable. The B has the parting line in the middle. The two cavities are shallow but have to be accurately made. If not, there will be a mismatch between them. This is not possible in A because the plastic is molded in one cavity and sealed off with a flat
back plate. In B you would be concerned about having the parts stick on the
ejection side. Since both sides are identical a slightly smaller taper would be put
on the knockout side. This would not be necessary in A.

Moreover, C and D show a similar selection of parting lines except that the
piece is turned on the 3/16 in. end. This gives a deep cavity harder to make and
maintain. If a 1° per side taper was needed for ejection, configuration C would
have a difference of 0.035-in. from maximum to minimum on the 3/16-in.
dimension. This contrasts with a 0.003 difference for this dimension using B.
The part could also be molded on the 1-in. side as shown in E and F. A, C, and E
could be parted on the diagonals.

The projected area is the net area of molded surfaces parallel to the platens.
Normally a minimum of two tons per square inch of projected area is required in
the clamping system to prevent the mold from opening. Many times the limita-
tions of a machine is its clamping capacity rather than its plasticizing capacity.
Parting lines A and B have a projected area of 0.5 in.², E and F have a projected
area of 3/16 in.², and C and D of 3/32 in.². Thus if the projected area (clamping
capacity of the machine) was the limiting factor, five times more parts could be
molded using parting line $C$ and $D$ than by using those of $A$ and $B$.

It is always more difficult to hold a dimension that crosses the parting line. Under the severe conditions of injection there is always the possibility that the clamping mechanism will move several thousands of an inch. This will increase the dimension controlled by the parting line accordingly. Therefore, if the 1-in. dimension is critical it would not be wise to use $C$ or $D$. Any of the other four would contain the 1-in. dimension within the cavity and produce parts to a closer tolerance.

Sometimes the location of the parting line is determined by the characteristics of the molding machine available. Consider molding a 10-in. deep cylindrical container (Figure 2-5a). Constructing the mold with the parting line shown in Figure 2-5b with the cavity attached to the stationary platen and the core opening on the movable platen, the following mold sizes are required. The cavity half would be 12-in. long, the core side 16-in. long, and the closed mold height 18-in. To eject the piece after molding the machine would first have to open 10-in. to get the cavity out of the core and another 10-in. to clear the piece and 2-in. so that the piece can be ejected. This requires a minimum machine daylight of
40-in. If the container was 0.100 thick and molded in polystyrene it would weigh approximately 8 oz. Most machines with this injection size do not have a minimum 40-in. daylight.

If the parting line was changed so that there were two cams (Figure 2-5c) the maximum daylight could be reduced to 30-in. If the mold was split as shown in Figure 2-5d with a removable core, the daylight requirement would be less than 20 in.

Changing the parting line may cause or permit other changes. Configuration B results in a tapered container, usually with walls of even thickness. In configurations C or D the option is added of having parallel outside walls (or any other taper).
The parting line selection would also affect the gate location. Mold $B$ must be gated at point $G$. If it were gated on the parting line entrapped air would prevent molding a complete piece. Mold $C$ can only be gated at point $G$. Mold $D$ can be gated anywhere along the 10-in. section. As we see later there will be distinctly different mechanical properties when the gating is changed from the side to point $G$. The projected area in molds $B$ and $C$ is about 13-in.$^2$. In mold $D$ it is 38.3 in.$^2$. Asthetic requirements might not permit mold $D$, because of the gate mark location.

Two Plate Molds

Figure 2-6 shows a schematic representation of the cross section of part of a regular two plate injection mold. The part being molded is a shallow dish, gated on the edge. There are temperature control channels in both backup plates and in the cores and cavities. Since there is significant insulation between two pieces of metal, the use of channels directly in cores and cavities gives better and more efficient temperature control than just cooling the $A$ and $B$ plates.

Note the support pillars which are anchored to the ejector housing (back plate) and support the backup (support) plate underneath the cores in the $B$ plate. A machine knockout bar is shown. As mentioned, they remain stationary, and as the moving platen returns, they stop the ejector plate. The mold opens on the parting line and the sprue puller, which in this instance is shaped like a $Z$, pulls the molded sprue and runner with it. The part design and molding conditions keep the plastic on the core. As the ejector mechanism works, the parts are pushed off the core and the sprue puller moves out if its hole, allowing the parts and runner to be removed or fall.

Three Plate Molds. Suppose the dish was deeper and could only be gated in the top center section. The mold could be constructed as a one cavity mold feeding directly from the sprue. There are other alternatives, one of which is shown in Figure 2-7 and 2-8. There are six cavities located in two parallel rows of three. One cavity and core is shown with other significant parts of the mold.

The difference between this type mold and the one illustrated in Figure 2-6 is that it separates between the runner plate and the pin plate (PL-1) as well as at the regular parting line (PL-2). This is called a three plate mold even though a third plate is not always used or a fourth plate may be added. The plastic is injected through the sprue bushing into the runner channel, which is trapezoidal, and tapered and cut into the runner plate with the wider face of the trapezoidal cross section facing the injection side. The plastic flows into the part through an auxiliary sprue bushing. While this can be machined directly into the runner plate and cavity, it is good practice to have a separate bushing so that it can be replaced or changed.
When the mold opens, the $A$ and $B$ plate moves together. Sometimes this will occur normally. Other times latching mechanisms are needed. The mold opens initially on parting line 1 (PL-1). This breaks the gate and leaves the runner attached to the pin plate because of the undercut pins ($A$) attached to the injection backup plate and extending into the runner. After the separation has occurred at PL-1, the mold continues to open, separating at PL-2. The molded pieces remain on the core. They are then ejected in the conventional manner by a stripper plate. The tie rod ($C$) pulls the cavity plate, which in turn pulls the pin plate through stripper bolt ($D$). The pin plate is limited in its travel by stripper bolts $B$. When it is moved forward opening at PL-3 (it could also be moved by...
Figure 2-7  Schematic drawing of a three plate mold in closed position.

latches, chains, ejector bars or cylinders), the runner is stripped off the undercut pins, and the plastic sprue is moved forward out of the sprue bushing. The runner fall down, can be removed by hand, an air blast or mechanical wiper. The pin plate always stays on its leader pins. They must be long enough so that the plates can separate far enough to remove the runner. It is good practice to support the pin plate on its own leader pins, attached to the injection backup plate. This will prevent it from binding on the main leader pins.

This type mold works very well provided the workmanship is of good quality and the components fully sized and adequately designed. If not, cocking and binding will occur relatively quickly on heavy molds. It is sometimes necessary to put an extra set of leader pins and bushings to support the A plate. These should not be used to line up the A and B plates. In other instances small leader pins and bushings are put into the A and B plates to assure good line-up and compensate for wear on the longer leader pins.

Another use for three plate molds is to mold parts on the plates that open on PL-1, Figure 7. This is an old technique used for molding records; it is not new in injection molding either (5a). Today they are called “stack” or “stacking” molds. Suppose one were to injection mold three 8 x 10 in. plaques directly over each other in a mold with the prerequisite number of plates in a stack mold. What clamping force would be required, based on a 3 ton/in.² of projected area
Figure 2-8  Schematic drawing of a three plate mold in open position.

design figure? The answer is $3(8 \times 10)$ or 240 tons, not $3(8 \times 10 + 8 \times 10 + 8 \times 10)$ or 720 tons. For each force there is an equal and opposing force, therefore, the force in the rear of the first and the front of the second plaque cancel each other out, as do the rear of the second and front of the third.

RUNNER SYSTEMS

The runner is the connection between the sprue and the gate. It is a necessary evil. It should be large enough to allow rapid filling and minimum pressure loss, but not so large as to require the cooling cycle to be extended for the runner to
harden enough for ejection. Most jobs will permit the runner to be reground and reused. Regrinding is expensive, wastes material, is a source of contamination, is a place for foreign material such as screw drivers and other metal parts to enter, and causes a probable lowering of the physical properties of the plastic.

There are various types of runners. The full round runner is preferable, because it has the smallest surface to volume ratio. When a runner has to be on one side only, the best compromise is a trapezoidal shape. Half round and rectangular runners should not be used. The runner should have no undercuts and be polished. This gives less turbulence in the flow and slightly faster filling rates.

The shape of the runner depends to some degree on the location of the cavities. It must be remembered that the initial surge of material will cool as it goes through the sprue and runner. Injection of this cooler material into the cavity tends to set up stresses. This should be avoided where possible by cold slug wells and runoffs. The design of the runner is an important factor in keeping the cooler material out of the cavity. Figure 2-9a shows a radially runnered mold, with

![Diagram](image)

Figure 2-9 Runner systems: (a) radial, poor design; (b) radial, good design; (c) standard design; (d) “H” design.
the runner running directly into the piece; \( b \) shows the preferred method, allowing the cooler material to continue to the end of the runner and warmer material to branch off into the cavities. Sharp corners should be avoided to reduce turbulent flow.

In multicavity molds it is essential for all cavities to fill at the same rate. If not, there will be packing and incomplete shots. Some people attempt to do this by a "balanced" or \( H \) runner system (Figure 2-9\( d \)). The theory is that if the plastic flows exactly the same length to all cavities, all will be uniform. Unfortunately this is not always so, as the mold is not usually the same temperature throughout. The heat history of the molding material is also not consistent. The gates still have to be balanced; therefore, the theoretical gains compared to a standard runner system (Figure 2-9\( c \)) may not outweigh the considerably larger runner system which has to be reground.

The size of the runners depend on the type of material and the size and thickness of the parts. As a general rule, the less viscous materials, such as the styrenes, approximately 5-in. or less from the sprue bushing, use from 1/8 in. to 1/4 in. full round runners. Runners occasionally go to 5/16 in. and very rarely over 3/8 in. Runners for viscous materials like acrylics and polycarbonates are usually about 3/8-in. full round. It is better to start with smaller runners as they are much easier to increase than decrease. If much experimentation is expected the runners should be cut into removable blocks.

**Hot Runner Molds**

The runners must be reground and reused in most operations, if possible. A logical extension of the three plate mold overcomes this and is called a hot runner mold (Figure 2-10). This mold has a hot runner plate, which is a block of steel heated with electric cartridges, usually thermostatically controlled. This keeps the plastic fluid. The material is received from the injection cylinder and is forced through the hot runner blocks into the cavities. Theoretically this is fine. Practically, hot runner molds take considerably longer to become operational initially when the mold is first tried. There are problems of temperature control, gating, balanced flow, and drooling. The greatest difficulty is to prevent the nozzle leading into each cavity from freezing. If the nozzle is too hot, the material will drool. If it is too cold, it will freeze. In multicavity molds, it is difficult to balance the gates (essential for proper molding) and still prevent either freezing or drooling.

The hot runner mold is especially susceptible to tramp metal, paper, wood, and other contaminants which will quickly clog the nozzle. Start up for each run is more difficult as freezing of one or two nozzles will pack or flash the other cavities.

Various gate controlling valves have been tried with varying success in the
nozzle of the hot runner mold (6). The most successful require temperature control on the heater around the nozzle and in the blocks surrounding them. They are comparatively expensive although they often can be used on other molds. The consideration for their use is usually economic.

Unless the job is long running or there are technical considerations, three plate molds are preferable. This is not to be construed a condemnation of hot runner molds. The author has and has seen many such molds that run and run well. When they do they are usually run automatically, are a pleasure to watch, and highly efficient (7, 7a, 11).
Insulated Runner Molds

A cross between a hot runner mold and a three plate mold is called an insulated runner mold. The gating system is very similar to that of a three plate mold except that the runners are very thick, at least 3/4 in. in diameter. There is no runner plate. The back-up plate and A plate are held together by latches. The material, usually olefins, is injected. The outside of the runner freezes but insulates the center permitting the core to remain fluid. This acts as a hot runner. If the runner freezes during start up the two plates are separated and the runner system removed. As soon as the runner reaches equilibrium, the latch is kept closed and the mold is operated that way.

The nozzles present similar problems to those of a hot runner mold. Usually a heated torpedo is inserted in each gate and kept on continually. The wattage of the cartridge heaters are selected by experience. The heat output can be reduced by using resistors in series. These are more difficult to start and operate than a three plate mold and are usually restricted to the olefins (8, 12).

MATERIALS FOR CAVITIES AND CORES

Steel is the most often used material for injection mold sets, cavities and cores. Beryllium copper and other nonferrous materials are also usable. Iron ore is reduced by burning with coke and on occasion, oxygen, in blast furnaces to produce pig iron. Steel is made from pig iron by removing its impurities in open hearth, Bessemer, or electric furnaces. Impurities are removed by various techniques including vacuum degasining. It is beyond our scope to discuss the technique of producing the quality steels required for molds. Such information is available from the literature and from the pamphlets and booklets of the steel companies.

When tool steels are deoxidized by silicone and aluminum to prevent gas evolution upon solidification, they are called “killed steel”. These pockets of gas leave voids which are not desirable in mold steels.

The steel is cast into ingots which require additional mechanical work. If this work is done above the critical temperature, it is called hot worked steel, and below is called cold worked steel. Cold worked steel is usually done at room temperatures and increases the tensile and yield strength and reduces the ductility. The ductility is a property which allows permanent deformation of the steel without facturing by stress in tension. Hot working increases the mechanical properties but is particularly valuable in increasing the ductility.

Steel is converted by either rolling or forging. In rolling, the steel is continuously passed through rolls reducing it to the desired size. Forging is done with a hammer forge or with more slowly applied hydraulic pressure in a forging press,
the latter method being preferred.

Steel is converted either singly, such as hot worked steel which is converted to the final size in one continuous operation, or in multiple conversions, where the steel undergoes a series of heating, forging, and cooling operations. The harder alloy steels are multiple converted.

Alloys are made by melting other elements into steel to affect its properties. A very brief summary of what each alloy does follows.

Manganese (Mn) is found in almost all tool and alloy steels. It is an excellent deoxidizer and hardens and strengthens the steel though less than carbon. It decreases the cooling rate, increasing the hardenability.

Silicon (Si) in the range of 0.2 to 0.35% is specified in all alloy steels for deoxidation. Increasing silicon content raises the critical temperature, increases the susceptibility to decarburization and graphitization, and when combined with other alloys promotes increasing resistance to high temperature oxidation.

Nickel (Ni) is used to improve the low temperature toughness of steel. It lowers the critical temperature lessening distortion in heat treatment. Nickel permits lowering the carbon content to achieve a given strength level thereby increasing toughness and fatigue resistance.

Chromium (Cr) improves the hardenability best of all the alloying elements and is used in air hardening steels. It improves the surface resistance to wear, promotes carburization and in high percentages reduces corrosion.

Molybdenum (Mo) increases the strength toughness, hardness, and hardenability of steel. It increases the machinability. It has a strong tendency to form stable carbides inhibiting grain growth and making the steel fine grained. This adds toughness. It also intensifies the effects of other alloys.

Vanadium (V), while expensive, increases strength, hardness, and impact resistance. It inhibits the growth of grains during heating, permitting a higher tempering temperature. It intensifies the effect of other alloys.

Tungsten (T) increases hardness, strength, and toughness.

Aluminum (Al) is a deoxidizer and a degassifier. It slows down grain growth and is used to produce fine austenitic grain size. It is used to the extent of 1% in steels to be nitrided.

Steels are designated by the American Iron & Steel Institute (AISI) and the Society of Automotive Engineers (SAE), whose designations for carbon and tool steels are the same. Four numbers are used to designate steels other than tool and die steels. A letter prefix designates the method of manufacture: C designates open hearth steel; B, Bessemer steel; and E, electric furnace steel. The first two numbers designate the type of steel. The last two numbers indicate the average carbon content in one one-hundredth of a percent. Hence a C-1045 steel would be an open hearth steel containing a 0.43 to 0.50% carbon (average 0.45%). Table 2-1 shows the series designation and the approximate alloy con-
Table 2-1  AISI Numbers and the corresponding alloying elements which cause their designation. The xx indicate the average carbon content of the steel in one one-hundredth of a percent.

<table>
<thead>
<tr>
<th>Series Number</th>
<th>Approximate % of Alloying Elements Designated by the Identifying Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>13xx</td>
<td>Mn 1.75</td>
</tr>
<tr>
<td>40xx</td>
<td>Mo 0.20, 0.25</td>
</tr>
<tr>
<td>41xx</td>
<td>Cr 0.50, 0.80, 0.95; Mo 0.12, 0.20, 0.30</td>
</tr>
<tr>
<td>43xx</td>
<td>Ni 1.83; Cr 0.50, 0.80; Mo 0.25</td>
</tr>
<tr>
<td>44xx</td>
<td>Mo 0.53</td>
</tr>
<tr>
<td>46xx</td>
<td>Ni 0.85, 1.83; Mo 0.20, 0.25</td>
</tr>
<tr>
<td>47xx</td>
<td>Ni 1.05; Cr 0.45; Mo 0.20, 0.35</td>
</tr>
<tr>
<td>48xx</td>
<td>Ni 3.50; Mo 0.25</td>
</tr>
<tr>
<td>50xx</td>
<td>Cr. 0.40</td>
</tr>
<tr>
<td>51xx</td>
<td>Cr. 0.80, 0.88, 0.93, 0.95, 1.00</td>
</tr>
<tr>
<td>52xxx</td>
<td>C 1.04; Cr. 1.03, 1.45</td>
</tr>
<tr>
<td>61xx</td>
<td>Cr 0.60, 0.95; V 0.13, 0.15 (minimum)</td>
</tr>
<tr>
<td>86xx</td>
<td>Ni 0.55; Cr 0.50; Mo 0.20</td>
</tr>
<tr>
<td>87xx</td>
<td>Ni 0.55; Cr 0.50; Mo 0.25</td>
</tr>
<tr>
<td>88xx</td>
<td>Ni 0.55; Cr 0.50; Mo 0.35</td>
</tr>
<tr>
<td>92xx</td>
<td>Si 2.00</td>
</tr>
</tbody>
</table>

tent of carbon and alloy steels.

The designation of special tool and die steels is found in Table 2-2. The P-1 steels are easily hobbed because of their low alloy and carbon content. Since they are water hardening steels they have a high tendency to distort during heat treatment. The P-4 and P-5 are a little more difficult to hob because of their chromium content. They are used because the chromium permits oil hardening and less distortion, as well as increased cores strength and resistance to wear. The P-2 and P-6 steels are more difficult to hob but give a tougher finished cavity than the other steels.

Steels for hobs are usually of the S-1, A-6, or 0-2 types. The H-13 and H-23 steels are used for hot hobbing beryllium. H-13 steel is commonly used for cores because of its low dimensional change in hardening and its good toughness. The P-20 steels are very tough and generally supplied in a prehardened condition of approximately $R_C30$. It can be carborized and hardened later. Other types of prehardened steels are supplied with hardnesses up to $R_C42$. The advantages of a prehardened steel evolves from the fact they do not have to be hardened. Hardening can cause distortion and on rare occasions cracking. It also increases the polishing time (14-16).

The 18% Nickel maraging steel is relatively new. It is delivered in a $R_C30$
Table 2-2  Tool and die steels (AISI) classification.

<table>
<thead>
<tr>
<th>Type</th>
<th>Prefix</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water hardening</td>
<td>W</td>
<td>Oil hardening</td>
</tr>
<tr>
<td>Shock resisting</td>
<td>S</td>
<td>Air hardening – medium alloy</td>
</tr>
<tr>
<td>Cold work</td>
<td>O</td>
<td>High carbon – high chrome</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>Air hardening – high carbon</td>
</tr>
<tr>
<td>Hot work</td>
<td>H 1 – H-19</td>
<td>Chromium base</td>
</tr>
<tr>
<td></td>
<td>H 20 – H-39</td>
<td>Tungsten base</td>
</tr>
<tr>
<td></td>
<td>H 40 – H-59</td>
<td>Molybdenum</td>
</tr>
<tr>
<td>High speed</td>
<td>T</td>
<td>Tungsten base</td>
</tr>
<tr>
<td></td>
<td>M</td>
<td>Molybdenum base</td>
</tr>
<tr>
<td>Special purpose</td>
<td>L</td>
<td>Low alloy</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>Carbon tungsten</td>
</tr>
<tr>
<td>Mold steels</td>
<td>P 1 – P 19</td>
<td>Mold steels – low carbon</td>
</tr>
<tr>
<td></td>
<td>P 20 – P 39</td>
<td>Mold steels – other types</td>
</tr>
</tbody>
</table>

hardness which results from air cooling from 1500°F. It can be precipitation hardened by ageing at 900°F for 3 hr. This gives an $R_C$ of 52 to 54. Because of the low hardening temperature there is practically no distortion (17).

Stainless steels are used in molds to eliminate the effects of corrosion. Corrosion can be caused by water or molding compounds, particularly PVC. By commercial usage they are defined as steels containing 11 1/2 to 20% chromium. Type 420 is most often used. It is magnetic and can be hardened by oil quenching to approximately $R_C$ 52. Inserts of stainless steel are valuable as they have lower thermal conductivity than tool steel, hence can reduce warpage due to nonuniform cooling of plastic.

**Hardness**

Hardness seems to be an easy concept but has not yet been adequately defined. It is probably not a fundamental property but a combination of work hardening, elasticity, yield strength, and tensile strength. Nonetheless it is an important measurement in mold building. It is commonplace in ordering a mold to specify the hardness of the cavities, cores, pins, and other components. The method most used in America is the Rockwell test method. There are ten scales. The one used for steel is the $C$ scale. The Brinell system is also used here and in Europe. Conversion tables are available in the Appendix. The Rockwell system is a measure of the difference of the depth of penetration of a ball into steel between an initial small load and a final large load. If the steel is too hard it becomes brittle. If too soft, it does not provide enough protection against damage and wear. A Rockwell reading of $R_C$ 50-55 will give good results. Steel this hard is difficult to machine even with carbide. It is easily worked by grinding or electrical (EDM) or
chemical removal equipment. The cavity or core is readily machined from steel in the soft condition as it comes from the steel mill. It is then hardened. Pre-hardened steel is a compromise between machinability, hardness, distortion, and wear (18).

The harder the steel, the greater its compressive strength. This should not be confused with the modulus of elasticity of steel which is 30,000,000 psi and is the same for all steels. This is a measure of the deflection of the steel by bending. Hardness does not significantly effect this at all. The choice of materials will. For example, beryllium used in molds has a modulus of elasticity of approximately 20,000,000 psi and will therefore deform elastically about one and a half times as much as steel under the same stress conditions.

Hardening

The hardening and hardness of molds, cores, and cavities should not be left to chance. The selection of steel, design of the piece, heat treating before and during the machining operations, and the type and specifications for heat treating have much to do with the longevity, usefulness, and trouble-free operation of a mold. A brief review of the hardening of steel is in order.

Iron has two crystalline structures. Alpha iron has the iron atoms at each corner of the cube and one in the middle. Gamma iron has iron atoms at each corner of the cube and one each centered on each of the six sides. Heating pure iron to 1670°F changes its structure from alpha to gamma. At 2535°F it goes back to the alpha form but is called Delta iron. At 2802°F iron melts.

Carbon is the important alloy in iron relating to hardening. Alpha iron contains very little carbon in solution. The solution of carbon or other alloying elements in alpha iron is called ferrite. Gamma iron holds considerably more carbon in solution. When it contains carbon or other elements in solid solution it is called austenite. When the carbon is not in true solution with the iron it forms iron carbide (Fe₃C), which is an extremely hard and brittle compound known as cementite. When the eutectic mixture of iron and carbon is cooled, alternate layers of ferrite and cementite precipitate out. This structure is called pearlite. Banite structure contains cementite needles in a ferrite matrix.

If the steel is cooled slowly enough to maintain the equilibrium, pearlite structures result which are soft structures. As the rate of cooling increases the cementite and ferrite precipitate out more quickly and the distance between the layers becomes smaller. The carbide is more dispersed, making it more difficult for the layers to slide over each other, which causes the steel to become harder. When austenitic steel is cooled very quickly (quenched) the carbide does not have time to separate out. This supersaturated carbon structure is called martensite. This is very hard and brittle and must be tempered before it can be used.

The minimum rate of cooling to form a fully martensitic structure is called the critical cooling rate. Cooling the steel more slowly will give a combination of pearlite and martensite. When, as mentioned, the steel is cooled slowly enough to reach equilibrium the composition is completely pearlite.

Steel is hardened only when martensite is formed. Therefore, the steel must
be cooled or quenched fast enough to prevent the formation of pearlite or bainite. The hardening behavior of steel is shown by TTT curves (derived from T for temperature, T for transformation, and T for time). Pure steel of the thicknesses used in molds does not have a high enough thermal conductivity to cool the inside quickly enough. To overcome this, alloying elements are added which in effect shift the TTT curves to the right.

The steels are quenched either in water (brine), oil which quenches more slowly than water, and air which is the slowest cooling medium. They are commonly classified as water hardening, oil hardening, or air hardening steel.

In large pieces, the surface can be converted to martensite while the core is still warm enough to be 100% austenite. Martensite formation increases the volume and cooling decreases the volume. Therefore, the outside surface is cool and contracting while the inside surface is expanding while converting from austenite to martensite. This causes stress in the steel which may result in warping or cracking. Obviously, the slower the cooling (air hardening) the less probability of either occurrence.

Because of this, the steel should be tempered immediately upon reaching approximately 150°F. To temper, it is heated to a given temperature below the critical range depending upon the grade of steel and the hardness desired. It is held there for 1 to 6 hr. It is then allowed to cool in air. Tempering removes the brittleness and reduces the hardness. The resultant degree of hardness depends on the temperature at which it is tempered.

*Annealing* is a term used for the heating and slow cooling of steel. It can remove stresses, change the toughness, hardness, and other physical properties.

The simplest form of annealing is stress relieving. The purpose is to remove the stresses generated in previous heat treatment, cutting, forming, or other operations performed on the steel. The steel is heated and soaked at about 1200°F and then allowed to cool in the furnace to about 930°F. It is then cooled in air. Steel should be stress relieved before hardening. If full annealing is required (to permit machining, for example) it is done by heating the steel to about 100°F over the upper limit of the critical temperature allowing it to soak fully and then cooling it in a furnace or other slow cooling medium. The cooling rate is shown by the TTT curve.

**Case Hardening**

Alloy steels will not form or hob as readily as a low carbon steel. Therefore, most hobbings are done in low carbon steels. To obtain a hardness suitable for use in plastic molds, elements have to be alloyed on the surface. This is called case hardening. A case hardened steel will have a very hard surface and a tough ductile interior. These hardened surfaces are required to give the high polish and long wear needed for injection molds. The most common method of case hardening is called carburizing. In pack carburizing, a steel container is lined with a commercial compound usually containing about 20% of metallic carbonate and charcoal. The steel to be hardened is packed into this compound and a cover put on the box. It is heated to 1550 to 1750°F, depending on the steel and the results required. The length of heating time is determined by the temper and the
depth of hardness desired. For example, in a particular steel carburizing at 1600°
for 28 hr gave a depth of the case of 0.070 in. This is approximately the depth
of case used in mold making. It permits the grinding and fitting operations to be
done without going below the hardened steel. The parts are then quenched in oil
and tempered for the desired hardness.

Nitriding

Nitriding is a process in which nitrogen is introduced to the outside surface of
the steel. It gives an extremely hard, wear-resistant surface which is retained at
elevated temperatures. A nitrided case is as hard at 750°F as it is at room tem-
perature. It can be heated to a 1000°F without losing its hardness when cooled.
This would immediately suggest its use for heating cylinders, plungers and bush-
ings.

Nitriding is accomplished by heating the steel in a controlled ammonia atmos-
phere between 950 to 1000°F. The processing time is measured in days. Since
the part is not quenched distortion and warpage are very low. There is a slight
dimensional growth for which allowances can be made. Parts which might be
difficult to harden by other methods are sometimes nitrided. Nitriding is best
done with a special alloy steel, Nitrallloy®, containing 0.85 to 1.20% aluminum.
The case hardness is in the area of Rc70. The case depth can vary from 0.007 to
0.080 in.

Design for Heat Treating

A mold and its parts should be designed with heat treating kept in mind. Nearly
all serious failures of hardened steel parts are caused by internal stresses. Since
molding develops repeated thermal and mechanical stress special care is needed.

There are a number of simple principles and practices which improve the des-
ign for heat treating. The major cause of cracking caused by heat treating is
sharp corners. Sharp corners are exceedingly dangerous to molds and MUST be
avoided. Fillets, radii, and tapers are preferred.

Avoid sudden changes of thickness. This prevents even heating and cooling.
Holes should be at least 1 1/2 diameters away from the edge. Water lines should
be kept 3/4 in. away from the mold surfaces. Ample lands should be provided.
Holes should be placed with as much metal around it as possible. It is sometimes
better to grind a slot after hardening, such as a key way. Assuming there is a 7X
9-in. plate with a 6 X 3-in. cutout on one side. It might be better to make two
sections which will be hardened and fitted separately. Deep drilled holes should
be avoided as they are difficult to quench. If there is any question it would be
well to consult the heat treater.

Steel Requirements for Cavities and Cores

Uniform structure and freedom from defects are obviously essential for a good
cavity. It is more than distressing to invest weeks of labor on a cavity or core and
find that there is a defect or flaw, making it unusable. It is, therefore, advisable
to only use steels specified for molds by the steel manufacturer.

Machinability, which directly affects cost, is an important factor. It is very
hard to rate. High machinability is sometimes produced by adding sulphur and
other nonmetallic elements. This reduces the polishibility as does large amounts
of nickel. Most moldmakers have "pet" steels which they rate as very machin-
able.

The steel should be able to take such surface finishing or polishing as required
for the application. It should also have the suitable corrosion resistance.

If the steel is to be hardened, its heat treatability and distortion must be con-
sidered. Since there is always a possibility of repair, either during or after mold-
making, it should be readily weldable.

The strength and toughness requirements of the steel depends on the nature
of the cavity and core. It is also related to the method of gating and venting.
When molding abrasive materials, such as glass filled thermoplastics, erosion re-
sistance should be considered. References about the selection of steels are 14,
15, 16, 19, 20 and 20a.

Surface Finish

The surface finish of the cavities and cores is an important specification. It will
affect appearance, ejectability, and cost. In many instances, the difference in
mold quotes is the different conception the mold makers have of the polish re-
quired. Heretofore in the molding industry surface finish specifications have
been merely descriptive, such as good polish, medium polish, and sandblast.
Mold surface specifications are available as well as strips of metal which are
finished to these specifications. A list of terms commonly used to describe sur-
face finish follows.

<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lay</td>
<td>The direction of the predominant surface pattern caused by the method of fabricating the metal. It might be tool marks or stoning marks.</td>
</tr>
<tr>
<td>Flaw</td>
<td>An irregular or infrequent surface mark such as a hole, scratch, and ridge.</td>
</tr>
<tr>
<td>Roughness</td>
<td>Relatively fine spaced surface irregularity in the direction of the predominant surface pattern.</td>
</tr>
<tr>
<td>Roughness width</td>
<td>The maximum width in inches, of the surface irregularity included in the measurement of roughness height.</td>
</tr>
</tbody>
</table>
Roughness height A major specification, measured in microinches* and specified in one of the following ways; maximum peak to valley height, average peak to valley height, average deviation from the mean surface (either Root mean square, rms or arithmetical).

When one number is used to specify the height or width of an irregularity, it indicates the maximum value. Any lesser will be satisfactory. When two numbers are used, they specify the maximum and minimum permissible values.

Waviness is a surface irregularity which has greater spacing than roughness. This might develop from work deflections, vibrations, heat treatment, or warping strains. The height and width are specified. It can be seen then that “polish” should not be the only specification. A roughness height of 2, 4, or 8 μ in. would give a very high polish, but if the waviness height is 0.008 or 0.010 the piece might well stick in the cavity or on the core.

Drilling and milling give surface finishes of 63 to 250 μ in., finish turning, broaching, boring, and reaming 32 to 125 μ in., grinding 32 to 63 μ in., and polishing down to 2 μ in., all measurements rms. Roughness heights are usually specified as 2, 4, 8, 16, 32, 50, 125, 250, and 500 μ in.

If the cost of finishing a 120-μ in. surface is 1, then 50 μ in. would be double, 16 μ in. triple and 8 μ in. quadruple.

Mold Polishing

The need for a properly contoured, polished mold surface is self-evident (20b, 20c). The penalties for neglect are sticking (which makes consistent mold cycles and automation impossible), scratching, and surface blemishes. A stuck piece wastes molding time in removal and reestablishment of the cycle, materials in purging, impairment of physical properties, and possible damage to the mold while the part is being removed.

Attention to the ultimate polishing throughout the manufacture of the cavity and core is profitable. If they are hobbed, the higher the finish on the hob, the easier the polishing on the final piece. If parts are to be hardened they should not be highly polished, as this changes the mold surface and causes flaws that will not come out in subsequent polishing. Heat treating must be done carefully to minimize scale (21, 21a).

After hardening, the scale (which is a deposit of ferric oxide) must be removed. This is done by alkaline reverse current methods, vapor blasting or immersion in ultrasonic pickling solution.

Molds are polished by using abrasives starting with the coarser grits for more rapid material removal. The abrasives that are used are stones, abrasive papers,

*Dimensions are often given in microinches (1 μin. = 0.000001 in.).
polishing compounds suspended in a liquid medium (carborundum or diamonds), and polishing compounds suspended in solid media. Each moldmaker has his own combination of polishing compounds and techniques.

Stoning and abrasive papers are usually done by hand, although reciprocating units are available. Compounds suspended in liquid or solid are applied by cotton or felt buffs, or bristle brushes. Figure 2-11 shows the core of a fish tank aerator being polished with a cloth buff attached to a drill press. Bar compounds used for metal polishing in other industries are useful. A fast cutting stainless steel compound, red rouge, and a chrome oxide green rouge are used. A hard wheel, using a stainless steel compound, will bring up a good polish. A soft wheel with chrome oxide green rouge will give a high luster polish.

The direction of polishing is important. If the polish is done in the direction of ejection, it is called draw polishing. Many times this will eliminate a severe ejection problem. As noted in the section on ejection (p. 128), high polish is not only not always desirable but can be detrimental. The selection and manufacture of a proper mold polish is a highly skilled art.

Steel is highly susceptible to rust, especially after polishing, and should be protected by a suitable rust preventative when not in use. Molds that are run cold should be brought to room temperature before being coated with a rust
preventative. If not, in humid air particularly, water will condense on the mold and cause rust damage. Using stainless steel or chrome plating the mold prevents water damage. Maintaining a proper polish is the responsibility of the molder (22-24).

Mold surfaces can be textured and patterned by photoengraving and etching (25).

Chrome Plating

The use of hard (industrial) chrome plating on forces and cavities is commonplace. When properly used, it offers significant advantages to the molder. The chromium will not rust, oxidize, or tarnish. It is comparatively inert to the chemicals and by-products of molding material except for those giving off hydrochloric acid. Even under those conditions it is superior to regular steel.

Its surface will not be affected by water or dampness. This means that polishing is no longer a large part of mold maintenance and that care to prevent rusting or corrosion of cavities and cores is eliminated.

Chrome plated surfaces are hard. At a thickness of about 0.0015 in. their hardness is about 1000 Brinell. It has a low co-efficient of friction. In most cases chrome plating will help mold release.

Chrome plating will not improve the surface underneath. If anything, it will accentuate blemishes, scratches, dullness, and pit marks. Therefore the preparation and polishing of the mold before plating is extremely important. Many platers specializing for the plastic industry have polishing departments and it is recommended that they do the final polishing on the steel.

There are a number of disadvantages to chrome plating. Once the plating is partially removed, either deliberately, or by damage or wear, it is virtually impossible to blend the remaining surface so that a mark will not show on the molded part. It will continue to peel and must be stripped and rechromed. The mold should be in good working condition with all vents ground in and changes made before chroming. The cost, too, of chroming is not insignificant. There is the possibility of hydrogen embrittlement particularly in heavier coatings. Stress relieving in hot oil at 350°F will help the condition.

Small hardened parts are plated approximately 0.0003 to 0.0005 in. thick. As the cavities get larger the thicknesses increase to a maximum of 0.005 in. In prehardened steel the thicknesses are from 0.001 to 0.003 in., in beryllium cavities 0.0005 to 0.003 in., in steel castings to 0.001 to 0.003 in. and in electroformed molds 0.0002 to 0.003 in. Within these limits chrome plating can be used to build-up the surface for making dimensional changes.

Beryllium Copper — Pressure Casting

The second major material used in cavities and cores is beryllium copper. It is an
alloy of copper containing approximately 2 3/4% beryllium and 1/2% cobalt. It is usually supplied to the mold maker as a pressure casting from a hob. It is also available in thin rods and bars. The material is excellent for molds and will give long wear and quality parts (26).

Beryllium copper is chosen because of its thermal conductivity, its resistance to rusting, and its method of fabrication.

An important function of the mold is to remove heat from the plastic. This is done by circulating a cooling fluid through the mold. Assuming identical molds with adequate cooling capacity the rate at which heat is removed from the mold will depend on the thermal conductivity of the metal. The thermal conductivity of beryllium is approximately two and a half times that of tool steel and four times that of stainless steel. Since the time required for cooling a plastic in the mold is a function of heat removal, a beryllium mold should give faster cycles if cooling is the limiting factor in the molding cycle. There are times when rapid cooling is not desirable or different cooling rates for different parts of the mold are required. If these factors are critical, they will form a major consideration in selecting the mold materials (27).

The surface of beryllium copper does not rust. Very often the refrigerated cooling media are circulated causing condensation on the molding surfaces if the cycle is interrupted. This will cause no mold problem in beryllium copper as compared to steel which will rust severely. A 0.0003 to 0.0005-in. flash chrome will appreciably increase the life of the cavity.

A major reason for using beryllium copper is because of the relative ease of fabrication using pressure casting, or more accurately hot hobbing (28, 29).

A hob is made proportional in size, but larger than the finished plastic part. (After hobbing the beryllium copper shrinks as it cools, just as plastic shrinks after molding.) The hob is placed in the bottom of an insulated cylindrical container. (For more on hobbing see the hobbing section, p. 119.) The melted beryllium copper is poured over it and a plunger comes down exerting pressure on the beryllium and hob. When the beryllium is cooled, the hob is separated. When casting cavities, use a shrinkage of 0.004 in./in. and on cores 0.008 in./in. is suggested. A steel hob is used made of a good hot working die steel such as H-13 or H-23 which will not deform under the temperature of casting. For one or two cavities, beryllium copper itself can be used for the hob, but as the number of cavities cast increases, the beryllium copper hob will tend to deform. A draft of 1½° per side is desirable though one-half of a degree can be used. The pressure cast cavities are reproducible with a tolerance of about ±0.002 in./in.

The nature of the process permits hot hobbing of thin sections and projections, intricate carvings, fluting, serrations, and other delicate shapes not hoppable in steel. The tensile strength of steel is not enough to prevent breaking of delicate sections under the high pressures of cold hobbing.

Hobs for pressure casting can be cast from an original sample model by a
number of casting processes. If there is an irregular parting line the cavity can be
cast from the hob and the wall thicknesses machined off the hob and then used
as a core. This gives a good parting line match and saves most of the machining
of the core.

Pressure cast beryllium copper cavities are usually supplied to the mold maker
annealed and hardened. The hardening range $R_C 35$ to 40 is soft enough to
machine but hard enough to perform well in long production molding runs. They have an ultimate tensile strength of 140 to 165,000 psi. The polishing tech-
nique is similar to that used for steel. The cavities can be silver soldered or elec-
ctric arc welded.

The cost of beryllium copper and steel cavities are similar. Cast beryllium
copper cavities are usually sold on a poundage basis. Its specific gravity is about
8.09 which is equivalent to 0.292 lb/in.$^3$.

Other Mold-making Materials. Other cavity and core materials used for
sampling or very low production molds (30) are brass, aluminum, steel filled
epoxy (31), and metal sprayed concrete shells (32). These are not materials of
choice for production runs or quality parts.

METHODS OF FABRICATING CAVITIES AND CORES

Most fabrication of metals for molds is done with tool room equipment. This
equipment is used for machining mold bases, cores, cavities, pins, blocks, and
such. More advanced equipment permits running of tool room equipment elec-
tronically by punched tape, which can be computer generated. For those not
familiar with tool room equipment, a very brief description follows.

1. A drill press is a tool that has a stationary table above which is a rotating
motor driven shaft. The shaft contains a chuck to which the drill or other tool is
attached. The shaft moves up and down. It can be hand or automatically fed.

2. A milling machine is a drill press with a table which can be moved left and
right, backwards and forwards, and up and down. The rotating shaft or spindle
in the head moves up and down. This is an indispensable tool. These movements
can be automatically or manually controlled. A separate attachment has a stylus
which moves to trace a three-dimensional replica of the part to be cut in steel.
As the tracer (stylus) moves in one direction the cutting tool moves in the same
direction with a proportional movement. This is now called a duplicator.

3. A lathe has a rotating head to which is attached the material to be cut.
The material rotates and the carriage, which contains the cutting tools, moves
along the length of the bed or across it. The tail stock is equipped with a chuck
for drilling and reaming.

4. A grinder has a rotating head to which is attached a grinding wheel. The
table reciprocates and can move in and out at a predetermined distance per reciprocation of the table. The height of the grinding wheel above the work is accurately adjusted. The grinder is used to obtain accurate dimensions and a good surface finish. It will readily grind hardened steel. Cylindrical grinders are used for grinding materials that can be held on centers and rotated. It is similar in concept to a lathe. External or internal grinding of round shapes is done.

5. Band saws consist of two wheels, one of which is motor-driven, upon which rotates an endless belt of saw material. The material to be cut is fed into the blade. Cut-off saws are movable while the material to be cut is held in a vise. There are two types; one is a band saw on a pivot and the other is a reciprocating saw which is mounted on a pivotong flame.

6. A shaper is used to cut and square blocks of steel. The steel is stationary and the head with the cutting tool reciprocates back and forth cutting on the forward stroke.

7. A planer does the same thing as a shaper except that the cutting head tool is stationary and the steel reciprocates.

8. A jig borer is a precision drill press with a table that moves similarly to a miller. Its purpose is to locate holes with extreme accuracy.

9. Welding equipment is of two types, electric and gas. In electric welding the heat required for melting is derived from an electric arc. In gas, by burning a combination of oxygen and acetylene. It is primarily used for repairing or changing molds and requires the services of a skilled technician. Welding is a common method of joining metal and is used on machinery and equipment throughout the plant.

Hobbing

Hobbing is the cold forming of metal. The term is used in plastics to designate the cold displacement of one material by another, caused by high pressures. For example, if a piece of plastic is left on a mold and the clamping pressure of the machine forces the plastic into the steel, the plastic is said to have hobbed itself into the mold. The term is used in mold making for the process which takes a hardened steel replica (hob) of the plastic part and by means of high pressure forces it into a soft iron block. Iron is very ductile. It flows around the hob giving an identical, but reversed impression. This is much the same as forcing a coin into a piece of clay. The steel selected for the hob must have good strength, a tough core and take a good polish. The hobbing blank, usually of soft Swedish iron, is carefully polished and fitted into the hobbing ring. This ring is a large piece of steel to contain the hobbing blank as the pressure of the hobbing press is applied. The iron of the hobbing blank slowly yields. In many instances the hobbing has to be annealed to relieve stresses and pushed again.

The design of the hob, the steel selected, its hardening, the amount of draft,
and the preparation of the hobbing blank are critical. If they are not correct, or the hobbing is not done properly the hob can crack.

Hobbing is a fast economical way to produce multiple cavities. All the cavities are the same size compared to each other and the hob. A high polish on the hob will be transferred to the cavity. Since the cavities are iron they must be carburized after they are machined to size. Figure 2-12 shows the hob for a plastic column. The hob is on the upper right. Beneath it is the molded plastic part which is identical in shape. The size of the molded parts will be smaller than the hob because the plastic shrinks in cooling. The upper left-hand piece is a hobbed cavity, and beneath it is the finished hardened, polished cavity, ground to size and ready to be put into the mold base (33, 34).

Casting

Recent technique for casting is so improved that this method is readily adaptable for injection molds (35, 36, 37, and 38). Any metal can be successfully cast, particularly with the Shaw process. In this patented process, a sample (or a plaster reversal) of the part in plastic, wood, metal, or other material is cast against a ceramic slurry. The slurry is fired and gives a reverse ceramic reproduction with a micrograin structure filled with small air gaps. The gaps act as vents so that the molten metal can achieve a good reproduction of the surface. The resultant cavity is not as dense as those produced by other methods, and there is the possibil-

Figure 2-12  (a) Hardened metal hob; (b) hobbed cavity; (c) hardened polished cavity finished to size; and (d) molded part. (Robinson Plastics Corp.)
ity of small pits. The appropriate shrinkage factors for the slurry, metal, and molding must be calculated. A new slurry casting must be made for each new cavity. The major advantages of casting are its speed and cost. A cavity can be made in less than a week. The economics depend on the size and the nature of the part.

**Electroforming**

Electroplating is old. It was tried unsuccessfully for mold cavities and failed primarily because the stress in the tool caused by the plating, caused the tool to deform in the molding process. This has now been overcome. A master, sometimes called a mandrel, is the exact reverse of the cavity. It can be made of any nonporous substance which will stand mild heat and upon which a conductive metal can be deposited. The deposition is usually a silver nitrate-silver system or vacuum deposition. The mandrel acts as the cathode.

A nickel cobalt compound is deposited at approximately 0.0005 in./hr to a total depth of about 0.150 in. Behind this is plated pure electrolytic copper at the rate of 0.003 in./hr through the thickness required for the cavity. The copper is harder than mild steel being approximately 220 Brinell. The deposition takes approximately 2 weeks for the nickel and 2 weeks for the copper. This is not excessive in terms of cavity production by other means. The polish of the mandrel can be reproduced with finishes in the neighborhood of 5 μin., so that cavity polishing is eliminated. For areas impossible to reach with polishing equipment, or where polishing will effect delicate lettering, this is invaluable.

Slots as deep as 3 in. with a opening of 1/4 in. on top are formable. For projections a hardened steel blade can be set in and the electro forming be done around it, sealing it permanently into the cavity.

If a number of cavities are made at once on the same plate they can be electroformed very closely together with the runner. This can increase the number of cavities in a given area as no side wall is required for support between the cavities.

The regular parting lines can be made to match perfectly. Since there is no hardening there is no heat distortion. The mold surface is noncorrosive. The cavity has exceptionally high thermal conductivity. Very high precision is possible.

Electroforming is used for such qualities as reproducing of fine surfaces, very close tolerances, and shapes difficult to fabricate by other means such as gears.

The process adapts itself well where accuracy is required. It is used, for example, for gear cavities, slide rule cavities, and intricate electrical connectors. Making cavities for external threads on plastic pieces is difficult by conventional machining. An electrode can be turned in a lathe and can readily form two matched cavities. Even though the process is entirely different, it can be considered to have the advantages of cold hobbing in hardened steel.
The surface finish is determined by the electrode and current density and frequency. The usual finish is approximately 10 to 20 μin., and can easily be polished by conventional methods. The quality of the finish is determined by economics and the physical shape of the parts (39, 39a).

**Duplicating**

Duplicating is mechanical reproduction by cutting tools which are guided by a master, proportional in size to the desired finished parts. Duplicating is mostly used for large parts, as hobbing and casting will usually reproduce a smaller one more economically. Large automatic duplicators are basically powerful horizontal millers with hydraulically controlled feeds. Using feedback and electronic techniques maximum cutting speed is obtained. Such processes as producing mirror images are easy. They can be automatically run and tape controlled. Small duplicators are often used in making hobs or engraving small designs, letters, and numerals on cavities. A major disadvantage of duplicating is its comparatively poor surface finish (40, 41).

**Erosion**

A new and useful method for removing steel is by electrical discharge machining (EDM) (42, 43 and 43a). An electrode, usually made of carbon although it can be made of any conducting material, is made in the reverse shape of the part to be produced. The steel and electrode are immersed in a circulating solution, which serves to flush away the eroded material and cool the work. AC power is rectified and charged into a capacitor system. This discharge between the electrode and the cavity creates a spark which erodes the steel. The electrode is eroded about one-eighth as fast as the steel. Roughing electrodes are used to bring the cavity to its approximate shape and a finishing electrode brings it to size.

The process is accurate, produces good detail, can be used with hardened steel so that no heat distortion takes place, and can be used for cutting thin slots. By eroding on one plate the distance between cavities can be reduced. Cutting is relatively slow. The preparation of the electrodes and the operation of the equipment requires an excellent toolmaker. Spark erosion is widely used in changing and correcting hardened steel cavities.

Chemical removal of steel is slowly being accepted in the mold-making industry (44).
GATES

The gate is the connection between the runner and the molded part (45-47a). It must permit enough material to enter and fill the cavity, plus the extra amount required to prevent excess shrinkage. The literature is full of articles relating to the size, type, and location of gates, and their effect on the molding process and the physical properties of the molded part. This section considers gate types and sizes. The effect of gates are discussed in the section on molding theory.

Gates can be classified as large or restricted (pin pointed). Restricted gates are circular in cross section and for most materials do not exceed 0.060 in. in diameter. The more viscous materials may have restricted gates as large as 0.115 in. diameter. An example of a large gate, which is usually square or rectangular, is 1/4 in. wide by 3/16 in. high. They are used for molding heavy sections and where the restricted gates give a surface blemish problem.

Restricted Gates

The restricted gate is successful because the apparent viscosity of the plastic is a function of the shear rate. The faster it moves, the less viscous it becomes. As the material is forced through the small opening its velocity increases. The shear rate is directly related to the velocity. In addition, some of the kinetic energy is transformed into heat, raising the local gate area temperature. Once the gate is opened to the point where it loses this shear rate effect (viscosity improvement), a much larger opening is required to get any reasonable flow. This is why there is a jump in size from a restricted to a large gate.

The size of the restricted gate is small enough so that when the flow ceases, the gate material will cool, sealing off the hydraulic pressure of the runner system from the cavity very quickly. This means that the flow of plastic must continue, once started, until the cavity is filled. If the plastic freezes at the gate there will not be enough pressure in the runner system to force the plugged gate into the cavity and resume the flow of material. Therefore, in multicavity, restricted gate molds, the gates must be balanced so that all cavities fill together.

Since the gate freezes quickly when the flow of material stops, there is little chance of packing the gate area. Gate packing is a common cause of overstressing plastic causing environmental failure. Since it will also prevent adding material to compensate for shrinkage, pieces molded with restricted gates show measurably higher shrinkages than the same piece molded with a large gate. When properly designed restricted gating will minimize packing and sticking in the mold.

One of the obvious advantages of small gates is the ease of degating. In most instances the parts are acceptable if cleanly broken from the runner. It also lends itself to automatically shearing the gates during the ejection part of the cycle. This is helpful in automatic molding. It is important that restricted gates come from full round runners. This permits the hottest material to enter the cavity
and not have the cold material, which would be dragged along the half runner, restricting the flow into the mold.

Before opening gates to correct molding conditions such as insufficient filling, sink marks, or bubbles, one should be sure the runner system is adequate. If there is insufficient transmission of hydraulic pressure due to the runner, the flow might slow down enough so that the gate freezes before complete filling. Also, low hydraulic pressure will slow down the rate of filling, reducing the velocity-viscosity effect. If a radical increase of injection pressure has little results on the piece, it is an indication of insufficient runner size.

When material from a restricted gate shoots into the mold it is in the form of a thin stream or jet. It shoots across the cavity, hits a wall, is cooled and folds over and over, creating a mass of cool material in the mold which is not always reheated and absorbed into the plastic. This can result in internal stresses, flow marks and surface blemishes. One way to overcome this is to mold against a tab or a pin or wall. Another way is to heat the mold section where the plastic hits. This can be checked by locally heating it with a torch. If effective a small cartridge heater under the gate area or a beryllium heat sink may do the job. If not the gate can be opened to a large size. Control of the injection rate so that a slow initial fill builds back pressure in the cavity before the major high speed fill is initiated will minimize this effect.

When molding heavy sections it is necessary to add a considerable amount of material to the place the volume lost as the plastic decreases in temperature. Small gates freeze quickly preventing material from entering. Large gates stay open much longer. Unless the machine conditions are correctly set large gates will permit the material to flow backwards out of the cavities and into the runner.

Restricted gates have the benefit of better mixing. It is virtually impossible to mold a good variegated pattern (mottle) without going through a large gate. Dispersion or mixing nozzles on the machine use the principle of the restricted gate. Many small restrictions are placed on a plate inserted between the nozzle and the cylinder. This restricts the flow as the effective cross section is much smaller than in an unrestricted nozzle.

Location of Gates

Gates are also described by location, such as edge gated, back gated, submarine gated, tab gated, and nozzle gated. Figure 2-13 shows examples of various types of gates. A sprue gate feeds directly into the piece from the nozzle of the machine or a runner. It has the advantage of a short direct flow, with minimal pressure loss. Its disadvantages include the lack of a cold slug, the possibilities of sinking around the gate, the high stress concentration around the gate, and the need for gate removal. Most single cavity molds of any size are gated this way.

Edge gating is most common. It can be the large type or restricted. If the
Figure 2-13  Different gating designs. (Robinson Plastics Corp.)
large gate is spread out, it is called a fan gate. If the gate is extended for a considerable length of the piece and connected by a thin section of plastic, it is called a flash gate (47a). Sometimes it is necessary to have the gate impinge upon a wall. This distributes the material more evenly and gives improved surface conditions. Walls are not always available. To overcome this a rectangular tab is milled into the piece and the gate is attached there. This is called a tab gate.

In gating into hollow tubes, flow consideration can require an even injection flow pattern. A single gate will not be sufficient. Four gates 90° from each other will often give four flow lines down the side of the piece which can be objectionable. To overcome this a diaphragm gate is used. The inside of the hole is filled with plastic directly from the sprue, and acts as a gate. It must be machined out later. A ring gate accomplishes the same thing from the outside.

A submarine gate is one that goes through the steel of the cavity. When the mold opens the plastic shears at the gate. A properly placed knockout pin, using the flexibility of the plastic, ejects the runner and pulls out the gate. This type gate is usually used in automatic molds.

Flow Through the Gates

Figure 2-14 shows a schematic drawing of a cavity block with half of a round runner and a gate. A gate is sometimes called a land; the gate or land length is important. The flow through an orifice is given in the generalized formula:

\[ Q = \frac{K \Delta P}{\mu} \]  \hspace{1cm} (2-2)

\[ \text{Circle } K = \frac{\pi R^4}{8L} \]

\[ \text{Slit } K = \frac{Wh^3}{12L} \]

where
\begin{align*}
Q & = \text{flow rate} \\
\Delta P & = \text{pressure drop} \\
\mu & = \text{viscosity} \\
K & = \text{geometric constant} \\
L & = \text{length of opening} \\
h & = \text{thickness} \\
R & = \text{radius} \\
W & = \text{width}. 
\end{align*}
Increasing the size of the opening ($R^4$ or $Wh^3$) has a major effect in the flow rate. However, this is counteracted by reducing the velocity which will raise effectively the viscosity. A point will be reached where small increments of $R$ will no longer increase the flow, but because of viscosity changes, will decrease the flow. This is the effect described previously comparing large and small gates. With very large increments of $R^4$ the effect of viscosity will be less important. Reducing the land length ($L$) will increase $Q$ and since it does not change the
cross sectional area, it will increase the velocity. This will have the added effect of reducing the viscosity. It also increases ΔP. For these reasons land lengths are kept to a minimum. Experience has shown that the shorter landed gates remain open for a longer period.

Ejection Systems

After the part is molded it must be ejected from the mold. Parts are ejected by K.O. (knockout) pins, K.O. sleeves, stripper rings, or stripper plates, either singly or in combination. The considerations for ejection are similar to those for parting lines. Additionally the quality of the molded piece is affected. We will not consider undercuts at this point. (An undercut is an interference by the mold to delay or prevent mechanical ejection of the plastic parts.)

The geometry of the parts and the plastic material are the major factors in selecting the knockout system. Most parts eject readily with a 1° per side taper. They can be ejected with smaller tapers but this should be done only if required. A high polish is not always required for easy ejection. The direction of the polish is more important. Draw polishing (stoning and polishing in the direction of ejection rather than randomly) is important in difficult cases. With some materials, such as the olefins and nylon, fine sand blasting may help. Normally a moderately polished surface will not present ejection problems.

The cross-sectional area of the knockout pins or rings must be large enough so that the knockout does not damage the piece. Aside from the obvious, when the knockout pins go through the molded parts, serious stressing can be caused in the knockout area. Birefringence studies of transparent molded parts show this clearly. It is desirable, although not always possible, to use large diameter knockout pins.

The molder and moldmaker usually are able to predict knockout problems. Often aesthetic considerations or lack of room for knockouts prevent using the number and size of pins desired. It is poor practise to build a mold under these conditions. If satisfactory ejection cannot be designed initially on paper, it will be difficult, if not impossible, to install same on the completed mold. Sometimes the parts have to be redesigned or made in two parts and joined later. Unless a mold can eject consistently, an even cycle cannot be maintained, and this part cannot be produced on a production basis.

The location of the cooling channel directly affects the location of the knockout pins. While it is possible, using “O” rings, to send a knockout pin through a cooling channel, it is certainly not desirable. Cooling channels should be designed away from knockout locations.

Figure 2-15 shows a pen barrel 6 in. long, 5/8 in. in diameter and closed at one end. The mold has sixteen cavities and was gated in the back. The core is slender and unsupported. An initial attempt was made to mold one cavity with the gate in the center (Figure 2-15a). This was unsuccessful because the pressure
Figure 2-15  Effect of gating on molded pen barrel.
of the molten plastic bent the core. When the mold opened the steel straightened out and scratched the plastic piece on the edge of the cavity as the platen moved back. This was overcome by gating each cavity in four spots (Figure 2-15b). This equalized the pressure around the core to the extent that it prevented noticeable bending and ejection problems. As an experiment all 16 cavities were singly gated at a later date. At the first full shot the mold froze closed. As the machine opened it stripped the threads of the clamps holding the mold to the platen. While this is an extreme case, it illuminates the problems that can be caused by the flexing of steel.

Figure 2-16 illustrates a condition known as entrapped material also caused by the elasticity of steel and its deformation under molding pressure. The item molded is a cup with an innercircular compartment. When the mold opens the molded piece remains on the core. The annular plastic ring A is entrapped between the core C and the annular steel ring B. This can be very difficult if not impossible to eject, particularly if A is thin. The proper way to build the mold is to have C retract in relation to the rest of the core before ejection.

![Diagram of a cup with an innercircular compartment showing the annular plastic ring A entrapped between the core C and the annular steel ring B.](image)

Figure 2-16 Use of retractable core to prevent ejection difficulties caused by entrapped material.
In molding deep parts it is sometimes necessary to vent the core. If not, the vacuum will prevent ejection. Figure 2-17 shows such a system. A vent pin which seats on a taper is held in place by a light spring. When the material injects, the

![Diagram of venting pin to break vacuum on core. Pin is held closed by spring and sealed by the pressure of the molded material. (Robinson Plastics Corp.)](image)

Figure 2-17 Use of venting pin to break vacuum on core. Pin is held closed by spring and sealed by the pressure of the molded material. (Robinson Plastics Corp.)
force of the material will seal the pin. As the part ejects, the pin will move up, against the spring pressure, venting the core. Air pressure can be used to help eject the part.

A cause of part sticking related to the elasticity of steel is a condition called packing. This results from excessive injection pressures on the material. Aside from elastically deforming the steel, sticking is caused by deeper penetration of the plastic into the pores of the mold. Packing is most common in multicavity molds where the filling rate is not balanced. Some cavities fill first, their gate seals off and the full injection pressure is concentrated on the remaining cavities. Balanced filling will cure this condition.

Once a mold is completed and the mold does not eject properly, a number of things can be done. The mold surfaces should be carefully inspected to remove burrs, scratches, pits, and undercuts. Possibilities of core and cavities shifting or misalignment should be eliminated. The part should be filled evenly. Increasing the curing time should be tried to be sure that the part is fully hardened. Packing should be reduced to a minimum. The plastic should be inspected to see if it has been degraded in the processing. Lubricants should be tried. The most common is a silicone aerosol spray. Different formulations are available for different materials (48). The stearates of the heavy metals are also excellent lubricants. Graphite impregnation of the steel is reported to greatly reduce ejection problems (48a).

If all the above fail, attention must be paid to mechanically changing the mold. Tapers should be evaluated. Locations for additional knockouts should be considered. Increasing the diameter of existing knockouts might help. The molder and moldmaker have many techniques to overcome ejection problems. Notwithstanding there are a few parts which cannot be ejected the way they are designed.

From a properly designed and built mold, parts should eject cleanly. If they do not, mechanical modifications should be made during the initial run to achieve this end. Lubricants, manual operation, unusual cycles, air, and such should be considered only as temporary solutions pending permanent mechanical solution of the ejection problem.

Figure 2-18 shows a stripper plate ejection system. The cores are stationary. Around them are hardened stripper bushings which are mounted in the stripper plate or plates. There is clearance in the lower part of the stripper bushings to minimize wear. The knockout bars cause the stripper plate to move in relation to the core pin leaving the part either on the plate or free to fall off. Figure 2-19 shows a stripper ring ejection system. The stripper bushings or rings are attached to the ejector plate and act as ejector pins as they move in relation to the core pin.

Sometimes it is necessary to eject in two stages. Figure 2-20 shows a double acting K.O. system. When the K.O. bar is activated the main K.O. plate, K.O.-1,
Figure 2-18 Stripper plate ejection system.

Figure 2-19 Stripper ring ejection system (Drawing reproduced with the permission of the National Tool, Die and Precision Machinery Association).
Figure 2-20  Double acting ejection system.
moves forward ejecting by means of ejector pins $A$. As the ejector plate continues to move it hits the second ejector plate, K.O.-2, which actuates ejector pins $B$. A separate push-back system is needed for returning the second K.O. plate.

Double ejection motion of a stripper plate followed by knockout pins is obtainable by mechanical methods utilizing a modified return pin (48b).

If hydraulic ejection is not available from the machine, a simple substitute can be used. The knockout plate should be extended beyond both sides of the mold. Two hydraulic cylinders should be attached to the extended plate, one on each side, with their rods attached to the moving platen. Activating the cylinders will be, in effect, equivalent to the hydraulic knockout action built into the molding machine.

In trying out new molds, parts often stick in cavities and cores. They should be carefully removed by a competent tool maker or molding expert using wood and soft copper. Occasionally the only way to remove the plastic without disassembling the mold is to heat a piece of metal with an undercut, such as a screw or part of a saw blade, and carefully insert it hot into the plastic. When it is cool, pulling the metal part may also remove the stuck plastic. On deep parts, blowing a jet of compressed air across the top, after the part has cooled slightly, may affect a release. Many times short shots do not eject because they do not mold over the knockout pins. If this tends to be a problem, extra pins for the short shot can be installed.

Cam Actions

Cam acting molds are common in the injection field. The cams are primarily used for molding parts with undercuts and holes whose cores, if left in place, would prevent ejection in the direction of the machine movement (49, 50). They are also used for engineering consideration relating to ejection, venting, and gate locations.

Cam action molds are more costly to build, more expensive to maintain, and require closer supervision during molding. It is sometimes possible to eliminate the cams. It might be possible to drill a hole or machine a slot after molding. Frequently this operation can be done at the machine without additional cost. Even if the work has to be done away from the machine it might be less expensive than a canned mold. This is particularly true of short runs.

Assembling of two or more parts can avoid cam actions. Again, this might be done at the machine. In some instances assembly can provide other benefits. Internal threads are easily formed by molding in two parts and assembling. Internal threads can be made by molding over removable inserts. They are unscrewed on the bench. Extra inserts are made so that the cycle can continue normally. This eliminates tapping plastic which should best be avoided. Tapping creates sharp edges which are focal points for breakage. This is called the
A "notch" effect. A metallic hexagonal nut can be readily located and held firmly by assembling it between two plastic parts, eliminating the necessity of an insert or tapping.

Figure 2-21 shows a sliding cam actuated by an air or hydraulic cylinder. It could also be run electrically through gear chains and sprockets although such mechanisms are mainly used in rotating unscrewing devices. This type of cam action is independent of the mold motion and may be actuated at any time in the cycle. This contrasts with mechanical cam action (Figures 2-22 and 2-23). The cam motions can be caused by the machine travel or the knockout system. Regardless of the activating mechanism, cams are held in place by some mechanical lock to prevent the injection pressure from moving the cam and putting excess stress on the cam bar. Sometimes it is necessary to have an ejector pin move in the path vacated by the cam. Unless the ejector pins are returned before the cam moves they will collide causing damage. To prevent this the

![Diagram of externally operated cam for molding hole in plastic. (Locking device not shown.)](image-url)
knockout plate should activate a switch which is tied into the machine circuit so that the cam cannot move unless the knockout pins are in place. If cams move so that the cam pin does not meet the hole, damage may result. It is sometimes necessary to use switches activated by the cam to prevent the machine closing if this happens.

Springs can be used to release undercuts. Figure 2-24 shows a cross section of an oval container with undercuts on its side. As the mold opens up the springs force the cavities up the angled slide releasing the parts. When the mold closes it forces the cavity down and compresses the springs. A mold of this design, producing a part 10 in. long, 4 in. wide and 4 in. deep, molder over 500,000 parts without any maintenance on the cam mechanism.
Figure 2-23 (a) Mechanical cam in ejection position (b) Mechanical cam in molding position. (Drawing reproduced with the permission of the National Tool Die and Precision Machinery Association).
Figure 2-24 Cavity slides up and out in the cavity block propelled by springs, until the undercuts are cleared. The plastic part stays on the core and is ejected normally.

Figure 2-25 shows a method of using K.O. pins for molding undercuts. They are called jiggler pins. The left-hand side shows a pin in the molding position. As the knockout plate advances the pin is pushed to the inside of the plastic part because of its configuration. It is anchored into the K.O. plate loosely so that it can slide laterally.

There are numerous other cam action systems. These actions are limited only by the ingenuity of the mold designer. Even though they are more expensive to build than conventional molds, one should not hesitate to use them if it
simplifies engineering of the part or the mold.

Internal or external threads can be molded automatically, or by using inserts in the mold which are removed with the piece and unscrewed on the bench. Internal threads can sometimes be made by molding one half of the thread (without cams or unscrewing actions) and letting the screw cut its own threads on the unthreaded side of the hole (53a). Extra inserts are used to save machine
time. Automatic unscrewing driving mechanisms include racks and pinions, gears, sprockets, electric motors, and hydraulic motors. Automatic unscrewing molds are considerably more expensive to build and maintain (51-54a). Thought should be given to using inserts in the mold (55-55a), adding metal inserts in a post molding operation, tapping the hole in the plastic, and using self-tapping screws.

VENTING

When the hot plastic is injected into the mold it displaces air. In a well-built properly clamped mold without vents, the injecting material may compress the air to such an extent as to prevent proper molding. The heat of compression in a nonvented mold might actually ignite the mixture of air and plastic, leaving a characteristic burn mark. Inadequate venting may also decrease the filling rate of the cavity, and in some instances prevent the cavity from completely filling.

The location and size of vents are still governed mainly by experience. Vents are first put in the obvious places before testing the mold. Additional vents are added as required.

Vents are usually ground on the parting line. The size of the vents depends on the nature of the material and the size of the cavity. A typical vent would be 0.001 in. deep and 1 in. wide. After the vent extended for a half inch the depth would be increased to 0.005 in.

Clearance between knockout pins and their holes provide venting. Sometimes special pins are placed in the mold just for venting purposes (Figure 2-17).

The gate location has a lot to do with venting and one is often restricted in gating because of the inability to completely vent the mold.

One way to decide whether venting is a problem in mold filling is to put enough brass shim washers over each leader pin so that the mold remains open approximately 0.005 in. If this does not cure the filling problem, its cause is not in venting. The ends of runners should always be vented.

Some parts cannot be adequately vented without vents so large that they cause objectionable molded edges. This can be overcome by machining a small tab in the desired location and connecting it to the molded part with a restricted gate.

Weld lines result from two hot masses of plastic joined in the part. Sometimes the only way to eliminate it is by molding a similar tab to remove the cold welded material.

Venting technique has been extended to the point where molds are evacuated by a vacuum to promote rapid filling and minimum degradation (56, 57).
TOLERANCE

A tolerance is the total permissible variation of size, form, or location. Tolerance should relate to the dimensional and performance characteristics which have to be met and kept for the effective and acceptable performance of the part. Tolerances should be specified for no other reasons.

Tolerance terms and dimensions are the method for describing the size of the finished piece. A definition of some common terms follow:

A *dimension* is a numerical value indicated on drawings with lines, symbols, and notes to define the geometrical characteristics of an object.

A *basic dimension* is a theoretical value used to describe exact size, shape, or location of a feature. It is a standard from which permissible variations are established by tolerances on other dimensions or notes.

A *form dimension* is one that specifies a feature which cannot be properly defined by dimensions of size or location, such as the angle of a thread or the angle of the frustrum of a cone.

*Location dimension* is one that specifies the position or distance relationship of one feature of an object with respect to another.

*Size dimension* is a specific value of a diameter, width, length, or other geometrical characteristics directly related to the size of an object.

A *reference dimension* is one without tolerances used for informational purposes and does not govern manufacturing or inspection operations.

*Datum, parts, lines, and surfaces* are features assumed to be exact for purposes of reference or computation and from which the location of the features may be established.

*Size* is a designation of magnitude. When a value is assigned to a dimension, it is thereafter referred to as the size of that dimension. The *actual size* is the measured size. *Basic size* is the one from which limits of size are derived by means of allowances and tolerances. The *design size* is that size from which the limits of size are derived by only the application of tolerances. If there is no allowance the design size and basic size are the same. The *nominal sizes* is the one used for general identification. For example, a rod may be referred to as a 1/4 - in. rod but the actual dimension is 0.248. In this instance 1/4 in. is the nominal size.

An *allowance* is the prescribed difference between the maximum material condition (MMC) of mating parts. It is the minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

*Unilateral tolerance* is a tolerance in which the variation is permitted only in one direction from the design size, (i.e. +0.000, −0.010). In *bilateral tolerance* the variation is permitted in both directions from the design size, (i.e., + 0.002, −0.006). In *limit dimensioning methods* the largest and smallest permissible dimensions are indicated. The tolerance is the difference between the two.
Fit is a term used to signify the range of tightness which may result from the application of a specific combination of allowances and tolerances in the design of mating parts.

The definitions above are taken from military standards (58) which is recommended for additional information.

In making drawings, fundamental rules, although obvious, are unfortunately not always followed. Enough dimensions should be shown so that the intended sizes and shapes can be determined without calculating or assuming any distances. At no time should one scale a print for dimensions. Each dimension should be stated clearly and be interpreted in only one way.

The purpose of a drawing is to elucidate. Not every one who uses a drawing is skilled in its interpretation. In complicated parts considerable thought and effort are required by even a competent draftsman. There is always a possibility of missing a vital spatial relationship. Since the plastic parts have drafts, parting lines, and perhaps undercuts, any method to simplify the understanding of a drawing is desirable. A simple isometric drawing, a perspective sketch or a brief description, can be helpful. A Polaroid® photograph of the part, enclosed with the drawing, is greatly appreciated.

When a part is molded initially all dimensions are inspected. For production purposes a limited number are checked. It is assumed that if the mold is not changed and certain specified dimensions are correct the remainder will be acceptable. It is important to determine which dimensions are to be inspected, the conditions of inspection, and the method of inspection.

The tolerances in mold making are relatively easy to define. Figure 2-26 shows the tolerance range of various types of machining. The relative cost of turning and finishing a steel casting is shown in Table 2-3.

It is evident that the higher the tolerance, the more expensive the mold. Costs rise geometrically. It is thus well to review mold dimensions and specify maximum tolerances.

<table>
<thead>
<tr>
<th>Table 2-3 Tolerance versus cost for machining a steel casting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operation</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>-------------------</td>
</tr>
<tr>
<td>Rough turning</td>
</tr>
<tr>
<td>Rough turning</td>
</tr>
<tr>
<td>Semifine turning</td>
</tr>
<tr>
<td>Fine turning</td>
</tr>
<tr>
<td>Grinding</td>
</tr>
<tr>
<td>Honing</td>
</tr>
<tr>
<td>Range of sizes</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>From</td>
</tr>
<tr>
<td>.000</td>
</tr>
<tr>
<td>600</td>
</tr>
<tr>
<td>1,000</td>
</tr>
<tr>
<td>1.500</td>
</tr>
<tr>
<td>2,800</td>
</tr>
<tr>
<td>4.500</td>
</tr>
<tr>
<td>7.800</td>
</tr>
</tbody>
</table>

**Figure 2-26** Machining tolerances for steel (MIL-STD-8B; 11-16-59).
Specifying mold dimensions is a function of the moldmaker and molder. One must estimate the shrinkage of the molded material, table 3-11. To some extent this is controlled by the molding conditions and the type and condition of the equipment. It is best to design to that side of the tolerance which, if violated, would require removal of metal from the mold. This is usually easy as contrasted to adding metal to the mold which at its worst might be impossible and at its best difficult. If hole or pin locations may be troublesome, they can be made by using larger pins in the mold turned down to size. They can later be removed, and new ones turned off center and relocated at a minimum cost. Consideration of molding and dimensional problems are mandatory and must be done in the design stage of the mold.

Plastic Tolerances

Tolerances are important because they prescribe the limits for the part. Ideally, these limits should be controlled by function and aesthetics. Many times they are controlled by ignorance or the desire to “play it safe.” These practises are costly in dollars, time, and customer satisfaction. Excessive tolerances are indicative of extremely poor engineering and should warn the molder and mold maker. They should go further into the application and see the functional and aesthetic requirements. Engineering drawings customer supplied notwithstanding, the part must function correctly. Ultimately the molder and the moldmaker will be faced with producing such a part. Even if they are paid for their extra work, the loss of their time and of customer satisfaction is serious. If the parts function properly and are aesthetically pleasing, they will almost invariably be accepted, print tolerances notwithstanding (59-61).

Suggested tolerances on molded parts in different plastics have been published by the Society of the Plastics Industry, Inc. (SPI). Figure 2-27 is for polystyrene. For a discussion of plastic tolerances see p. 275. If finer tolerances are required, the rejection rate will rise significantly and the part should be priced accordingly (62).

It is not too difficult to build a mold to tolerance. The tolerance limits of the mold-making techniques are accurately known. They can be used to produce a mold meeting the specifications of the drawing.

Selecting the right cavity size is essentially an educated guess. As the material cools, the volume shrinks. The shrinkage range which is specific for each material is given in inches per inch. These are what normally would be expected in molding. For example, polycarbonate has an expected shrinkage of 0.006 in./in. If a finished dimension of a part was to be 6.000 in., the cavity size would have to be $6 \times 0.006$ in. plus the shrinkage of the 0.036 in. (0.002) or 6.038 in. (63) (see p. 284).

The reproducibility of the molded part or the tolerance to which a part can be held will depend, among other things, on the shrinkage variation from shot to
**STANDARDS AND PRACTICES OF PLASTICS CUSTOM MOLDERS**

**Engineering and Technical Standards POLYSTYRENE**

**NOTE:** The Commercial values shown below represent common production tolerances at the most economical level. The Fine values represent closer tolerances that can be held but at a greater cost.

<table>
<thead>
<tr>
<th>Drawing Code</th>
<th>Dimensions (Inches)</th>
<th>Plus or Minus in Thousands of an Inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>A := Diameter (see Note #1)</td>
<td></td>
<td>1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28</td>
</tr>
<tr>
<td>B := Depth (see Note #3)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>C := Height (see Note #3)</td>
<td></td>
<td>6,000 to 12,000 for each additional inch add (inches)</td>
</tr>
<tr>
<td>D := Bottom Wall (see Note #3)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E := Side Wall (see Note #4)</td>
<td></td>
<td>007 002 0035</td>
</tr>
<tr>
<td>F := Hole Size Diameter (see Note #1)</td>
<td></td>
<td>0.000 to 0.125 002 001</td>
</tr>
<tr>
<td></td>
<td>0.125 to 0.250 002 001</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.250 to 0.500 002 0015</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.500 &amp; Over 0035 002</td>
<td></td>
</tr>
<tr>
<td>G := Hole Size Depth (see Note #5)</td>
<td></td>
<td>0.000 to 0.250 0035 002</td>
</tr>
<tr>
<td></td>
<td>0.250 to 0.500 004 002</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.500 to 1.000 005 003</td>
<td></td>
</tr>
<tr>
<td>Draft Allowance per side (see Note #5)</td>
<td></td>
<td>11/2 1/2</td>
</tr>
<tr>
<td>Flatness (see Note #4)</td>
<td></td>
<td>0.000 to 3.000 007 004</td>
</tr>
<tr>
<td></td>
<td>3.000 to 6.000 013 005</td>
<td></td>
</tr>
<tr>
<td>Thread Size (class)</td>
<td>Internal 1 2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>External 1 2</td>
<td></td>
</tr>
<tr>
<td>Concentricity (see Note #4) (T I R)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fillets, Ribs, Corners (see Note #6)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Surface Finish (see Note #7)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Color Stability (see Note #7)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**REFERENCE NOTES**

1. These tolerances do not include allowance for aging characteristics of material.
2. Tolerances based on 1/4" wall section.
3. Parting line must be taken into consideration.
4. Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is impossible to achieve.
5. Care must be taken that the ratio of the depth of a cored hole to its diameter does not reach a point that will result in excessive pin damage.
6. These values should be increased whenever compatible with desired design and good molding technique.
7. Customer-Molder understanding necessary prior to tooling.

---

Figure 2-27 Molding tolerances for polystyrene (The Society of the Plastics Industry Inc.).

shot. In polycarbonate, for instance, which has very high reproducibility, a close molding tolerance can be held. In this instance 6.000 in. plus or minus 0.010 in. is realistic. In polypropylene 6.000 in. plus or minus 0.025 in. is a realistic tolerance. The reproducibility or the amount of tolerance to which a material
can be held is an inherent function of the molecular structure, molecular size and molecular weight distribution of the polymer, which affects the volume-dimen-sional change in cooling from a hot fluid to a cold solid.

Given a cavity of a specific size there are many things which will cause variation in dimensions. The gate size, the gate location, the number of gates, and the size of the runners will change dimensions. By the time the mold is ready for production they will have been optimized. The other variables are inherent to the molding process. These include mold temperature, clamping pressure, material temperature, injection speed, injection time, injection pressure, overall cycle, and variations in materials. We shall subsequently see during the discussions of molding theory that these variations can only be predicted in general terms. The selection of proper mold dimensions is one of the more difficult phases of mold design.

Since these molding variables are compounded by uneven cycles caused by the operator, the tolerances should be ideally set so that these variations would still mold acceptable parts. When this is not the case, some method of control is required. The molded parts shrink for about 1 day with the major shrinkage occurring within the first hour of molding. Gages can be developed or measurements designed so that inspection information obtained soon after molding can be translated into acceptable or rejectable parts. This is one reason why close tolerances are particularly expensive in injection molding.

Sometimes shrink fixtures are used to maintain dimensions. In other instances rapid chilling of the molded parts, usually in water, will help maintain sizes. Consideration should be given to salvaging out of tolerance parts. Sometimes a simple machining or a post-forming operation would be economical.

WATER AND MOLD TEMPERATURE CONTROL

Reduced to its ultimate fundamental, injection molding is really the controlled heating and cooling of plastic. This section is concerned with the cooling or removing of heat from the plasticized material. Cooling of the molding machines is also briefly discussed.

Water is the universally used heat transfer medium in its temperature range. The equipment manufacturers will recommend chemical systems for use outside of that range which are compatible with their product. Although we generally consider that cooling water is readily available, ecologists warn that the present supply and distribution of potable water is not sufficient for the world's or even the community's predictable needs. Consequently many municipalities require treatment and reuse of waste water particularly in such applications as molding.

Furthermore, an efficient molding system uses a great deal of water so that recirculation often makes good economic sense. Recirculating can also yield
additional benefits in water quality and productivity.

A typical 16-oz molding machine uses approximately 300 gal or 40 ft\(^3\)/hr. Assuming 6000 hr of operation a year, it will use 240,000 ft\(^3\). Based on a typical water and sewerage cost of $2.00/1000 cubic feet, cooling the machine alone (not including the mold) costs $480. If the water was recirculated using a cooling tower the cost would be less than $100. With water rates spiraling (New York City’s is $4.00/1000 ft\(^3\)), economics and good (industrial) citizenship, therefore, dictate recirculating systems for molding plants.

**Basic Water Cooling Theory**

The standard unit for measuring heat is the British Thermal Unit (Btu); 1 Btu is the amount of heat necessary to change the temperature of 1 lb of water 1°F. The unit of refrigeration is a commercial ton of refrigeration which is defined as the removal of heat at the rate of 200 Btu/min, or 12,000 Btu/hr. The standard ton of refrigeration is 288,000 Btu, or the amount of Btus removed by a commercial ton of refrigeration in 1 day. It should be noted that the standard ton has the dimensions of heat while a commercial ton has the dimensions of heat divided by time. Table 2-4 gives some useful data for cooling calculations.

The branch of physics devoted to measuring the thermodynamic properties of moist air is known as psychrometry. Cooling towers, which are used in most plants, operate on psychrometric principles.

When a liquid changes its state to a gas (evaporation), it requires energy to loosen the molecular bonds. It receives this energy in the form of heat, taking the heat from the surrounding substances, thus lowering the temperature. This heat is called the *latent heat of vaporization* or the latent heat. Thus when water evaporates, it will remove heat from the surrounding water and air, lowering their temperature. This is the theoretical basis for cooling tower action. Conversely when gas is condensed, as in refrigerating systems, energy in the form of heat is liberated and the surrounding substance (the condenser) heats up.

**Sensible heat** is the heat required to change the temperature of the air or water without changing its state. It is a measure of the internal kinetic energy and changes with the absolute temperature of the body. The *latent heat* is potential energy and shows itself in changes of the physical state of the body (evaporation, condensation) and is not accompanied by any changes of temperature. The latent heat of evaporation of water (Btu/lb) at 60°F is 1059.3, at 80°F is 1048.1 and at 100°F is 1036.7.

The sources of cooling water are

1. Rivers and lakes
2. Cooling ponds and wells
3. Spray ponds
4. Government water supplies
5. Cooling or evaporative towers
6. Mechanical refrigeration
### Table 2-4 Useful data for cooling calculations

<table>
<thead>
<tr>
<th>One standard ton 288,000 Btu</th>
<th>One commercial ton</th>
</tr>
</thead>
<tbody>
<tr>
<td>removes 200 Btu/min</td>
<td>12,000 Btu/hr</td>
</tr>
<tr>
<td></td>
<td>288,000 Btu/day</td>
</tr>
<tr>
<td>cools</td>
<td></td>
</tr>
<tr>
<td>20 lb of water</td>
<td>10°F/min</td>
</tr>
<tr>
<td>200 lb of water</td>
<td>1°F/min</td>
</tr>
<tr>
<td>6 gal</td>
<td>4°F/min</td>
</tr>
<tr>
<td>12 gal</td>
<td>2°F/min</td>
</tr>
<tr>
<td>24 gal</td>
<td>1°F/min</td>
</tr>
</tbody>
</table>

Cooling 1000 gal of water 25°F requires removal of 208,000 Btu

- 1 gal = 231 in.³
- 1 gal = 0.1337 ft³
- 1 gal/min = 8.0208 ft³/hr
- 1 ft³ = 7.42 gal
- 1 ft³/min = 0.1247 gal/sec
- 1 ft³ of water = 62.43 lb
- 1 lb of water = 0.016 ft³
- 1 lb of water = 0.1198 gal
- 1 gal of water = 8.345 lb
- 1 Btu = 778.2 ft lb
- 1 hp = 0.7068 Btu/sec
- 1 kw = 0.9478 Btu/sec
- 1 Btu/sec = 1.055 kw

**Rivers and Lakes.** These sources are rarely available to molding plants. They probably will require filtering, settling, or chemical treatment. A constant check is required because of the possibility of upstream pollution and other changes. The use of this water usually requires the permission of local authorities.

**Cooling Ponds and Wells.** Aside from the sources just mentioned the cheapest method of cooling water is by means of a cooling pond. This is a pond of water where hot water enters on one side and cold water is removed on the other side. While it is inexpensive it is also inefficient and often unsatisfactory for molding plants. It has a low heat transfer rate and needs a large size. The cooling depends on air temperature, relative humidity, wind speed, and heat gain from the sun. During summer in middle northern latitudes the minimum water temperature expected would be about 86°.
Molds

If the pond is used for cooling and hydraulic system of machines and refrigerating equipment, the size of a pond to handle 1000 gal/min would be approximately 60,000 ft². The pond has to be protected from children, algae, bacteria, and such other contaminators that may fall on an open body of water. Needless to say the water must be chemically treated and filtered. Well water can be used if the supply is adequate and the temperature low enough.

Spray Ponds  An improvement over cooling ponds is a spray pond, which is a body of water over which a spray system is installed. The nozzles are approximately 8 ft above the water level. By presenting a much larger area for evaporation more cooling will occur. The disadvantages are those of an open water system, dependence on wind velocity, plumbing the spray, relatively high water losses, and possibility of the spray being a public nuisance.

Cooling Towers.  An important source of water cooling in the molding plant is the cooling tower. There are two types, one that depends on prevailing winds (atmospheric) and the other that depends on a forced air feed by fans (mechanical) which is used in molding plants. When the fan is placed on the bottom of the tower, it is called a forced draft tower and when on top it is called an induced draft tower.

This type of tower has the advantage of a small ground area per unit cooling. It can be located anywhere including inside loft buildings; it requires low pumping head; it can control the temperature of the water more closely than of any of the previously mentioned systems; and it is economical in terms of water consumption. Its main disadvantages are that it has a high operating cost primarily for the air circulating fan. Its maintenance costs are comparatively high; it is subject to mechanical failure, and can present problems in removing the hot, moisture-filled exhaust air.

The hot water is pumped into the top of the cooling tower from where it falls by gravity over a grid, cooling itself during its fall. It is collected in the bottom or basin from which it is pumped out as the cooled water for the processing system. The cooling is done primarily by evaporation. It is estimated that between 10 and 20% of the cooling is done by convection heat transfer between the cool air and warm water. The water drops on slats to break its fall. The amount of cooling depends on the length of time the drop of water is exposed to evaporation. Since falling bodies are accelerated by gravity the slats effectively decelerate the droplets. Additionally it breaks the "Thomson effect," which hinders evaporation because of a difference in electrical potential at different points on the sphere of water, caused by surface tension effects. The slats act to break up the flow of water and form new drops, thus giving a larger surface area and breaking this thermal barrier at the surface. They are made either of redwood or polyethylene.

The cooling range of a tower is the difference in temperature between the water intake and outlet. The heat load of a tower is the number of Btu per
minute removed by the tower. The circulation rate is the amount of water going through the tower per unit time. The heat load is the product of the circulation and the range. The amount of heat removed by the tower can be increased by the increasing of the area over which the water flows, increasing the amount of air flowing per unit time (velocity), raising the inlet water temperature and reducing the humidity of the air.

The principle involved in a cooling tower is the removal of sensible heat because of the difference in air and water temperature and the removal of latent heat by the change of state from water to water vapor of a small amount of the fluid. It takes approximately 1000 Btu to evaporate 1 lb of water. This is the amount of heat required to cool 100 lb of water 10°. Therefore for each 10° of cooling approximately 1% of the water circulated must be evaporated. This water plus the spray loss (tenths of a per cent) has to be replaced and is called the makeup rate.

All water contains dissolved chemicals. These are brought into the tower during makeup while pure distilled water departs during vaporization. Therefore in time the chemical composition of the water will cause scale and other problems in the molds and coolers. Water must be treated and filtered. The large velocity of air will cause dust and other particles to collect in the tower requiring cleaning. In northern climates during winter there is a possibility of the tower icing. This is overcome by simply bypassing some of the hot inlet water into the basin.

**Mechanical Refrigeration**

The mechanical refrigerator, like the cooling tower, removes heat from the system. The practical difference is that the heat removed by the cooling tower cools the water to temperatures which are controlled by atmospheric conditions. The heat removed in a mechanical refrigerator is removed at a temperature based on the design of the machine. For example, heat can be removed to cool water to 40°F while having the removed heat dissipated at temperatures of 85 to 100°F. Mold temperature control is a basic requirement for accuracy and economy in molding. The temperature of the mold should be determined by the optimum molding conditions, not by the available temperature of the cooling water. For this reason mechanical refrigeration is required in a molding plant.

The simplest mechanical refrigeration system would be a closed box containing an open dish of a low boiling chemical (refrigerant), a fan, and a vent. If the refrigerant were liquid ammonia it would evaporate at minus 28°F at atmospheric pressure. One pound would absorb 589.3 Btu in evaporating (latent heat of evaporation). If the temperature surrounding the box is above minus 28°F, the heat absorbed by the ammonia in evaporating would come from the surrounding media. This is the same theory as water evaporating in a water tower. This method is not practical for many reasons. If the refrigerant were
ammonia the odor and toxicity would make it unusable. The cost of the refrigerant would make it uneconomical. The technique would not allow for adequate temperature control. To overcome this the refrigerant is evaporated and mechanically condensed in a closed system and continually reused.

If the pressure is increased, the temperature at which the ammonia (refrigerant) will evaporate and condense is raised. At 47.6 psig, the temperature of vaporization is 32°F, at 92.9 psig it is 60°F, and at 197.2 psig is 100°F. Thus by changing the system's pressure the temperatures at which the change from liquid to vapor or vapor to liquid can be controlled. This means, in effect, that heat can be removed from the system at any convenient cooling temperature without depending on the atmospheric temperature as is required in a cooling tower. Machines designed for air-cooled operation are about 15% less efficient. For units up to 5 tons the convenience is worth the extra cost. These are mainly on portable units. For larger sizes water cooling is preferred.

In mechanical refrigerators the liquid refrigerant is charged into the receiver. The compressor is started. When the system operates, the gas is compressed. The condenser section removes heat from the vapor (either by air or water cooling) causing it to condense into liquid which is stored in the receiver, still under pressure. In large units the heat from this section can be used to help heat the plant. The gas is now expanded by an automatically controlled throttle valve which reduces the pressure so that the liquid refrigerant will evaporate. It does so in the evaporator absorbing heat from the surrounding environment. This causes the cooling. It then goes to the compressor where it is compressed again and the cycle repeated. In water cooled mechanical refrigerators, 2.4 gal/min per ton of refrigeration of cooling water are required for each 10°F of cooling. This is a good approximation for preliminary estimates, as many cooling towers used for injection molding operate at about that range during the summer months. The most common refrigerant is Freon F-12® (dichlorodifluromethane).

A mechanical refrigerator or a chiller is designed for outgoing water of a specific temperature. The most common is 50°F. Any deviations from this will change the efficiency of the unit. For example, water leaving at 60°F would raise the efficiency to 120%, 40°F would reduce it to 80%, and 30°F to 60%. Therefore a 10-ton chiller designed for 50°F would deliver 12 tons at 60°F and 6 tons of refrigeration at 30°F.

The refrigerated water can be supplied either from a central system or portable coolers for each machine. The main advantages of the central system is low initial cost and freeing floor space around the molding machine. It has a number of disadvantages. It provides water at one temperature requiring elaborate mixing systems for mold temperature control. It is relatively inflexible in terms of capacity. At the initial installation one has to guess the cooling requirement for the future. Individual chillers can be bought as required. Molding can be scheduled for their maximum utilization. For most custom
molding plants portable chillers are preferable (64).

Cooling Requirements

Two convenient equations for determining cooling loads follow:

\[ \text{ton} = \frac{\text{Btu}}{(12,000)} \quad (\text{hr}) \quad (2.3) \]

\[ \text{gal/min} = \frac{(\text{tons})(12,000)}{(\Delta t)(60)(8.3)} = \frac{(24)(\text{tons})}{\Delta t} \quad (2.4) \]

Molding machines are usually cooled with tower water. If the temperature-humidity conditions are too severe mechanical refrigeration is added as required. Tower water is much more economical and should be used when possible.

The heat exchanger of a 16-oz 400-ton hydraulic clamp molding machine with 45 connected horsepower was instrumented to determine the heat removed. This averaged 25,000 Btu/hr. It was relatively independent of the cycle time and ambient temperature. This is not indicative of the total heat loss as the machine radiates a considerable amount of heat energy. The cooler would require approximately two tons of refrigeration (Eq. 2.3). A convenient approximation of machine cooling requirement is 1 ton/20 connected hp. If a water tower with a 6° approach was used the machine would require approximately 8 gal/min of water (Eq. 2.4).

Mold cooling requirements are relatively easy to estimate. The enthalpy of plastics is a measure of their heat content and given in Btu per pound. Graphs are available of enthalpy versus temperature. In crystalline material they include the heat of fusion. By subtracting the enthalpy at room temperature from the enthalpy of the material at the cylinder temperature the number of Btu to be removed is obtained. Table 2-5 shows this for some thermoplastics.

These figures do not actually describe what occurs. When the molded part is removed from the mold, a considerable amount of heat is still in the part, which cools in the air. There is a significant radiation loss from the mold itself. The author molded a plaque of general purpose styrene 7 in. X 3 in. X 0.150. The heat loss through the mold water cooling was measured. The molded part was put in a calorimeter and the residual heat measured. The enthalpy graph of this particular material showed a heat content of 140 Btu/lb between molding and room temperature. There was 38 Btu/lb removed by the cooling water, 57 Btu/lb remained in the molded part and the balance of 45 Btu/lb was radiated from the mold. About 80 lb/hr were molded. The amount of refrigeration required is 38 X 80 or 3040 Btu/hr. This is approximately one fourth of a ton. Using the enthalpy from the graph, 140 Btu/lb, one would expect that a ton of
refrigeration would be needed. Practically the amount would vary with the geometry and thickness of the part and the size of the mold. Using 50 to 75% of the figures in Table 2-5 will give a good approximation of the required cooling (65).

Table 2-5  Enthalpy difference or heat content (Btu/lb) of some thermoplastics between approximate molding temperatures and room temperature

<table>
<thead>
<tr>
<th>Material</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polystyrene</td>
<td>155</td>
</tr>
<tr>
<td>Acetate</td>
<td>180</td>
</tr>
<tr>
<td>Acetal</td>
<td>180</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>210</td>
</tr>
<tr>
<td>Low density polyethelene</td>
<td>260</td>
</tr>
<tr>
<td>Nylon 6</td>
<td>270</td>
</tr>
<tr>
<td>High density polyethelene</td>
<td>310</td>
</tr>
<tr>
<td>Nylon 6/6</td>
<td>340</td>
</tr>
</tbody>
</table>

Mold “Heating” Units

The function of the fluid circulation through the mold is to control the rate of heat transfer, hence the cooling rate at the plastic. Elevated temperatures are used when slow cooling is required.

The temperature of the cooling medium will depend on the molding requirements. When the cooling medium is above room temperature, requiring the addition of heat, it is commonly called a mold heater, even though it is in effect cooling the mold. A mold heater is, in essence, a tank with a motor driven centrifugal pump recirculating a fixed amount of fluid from the tank through the mold. Adding heat to the fluid is done by electrical resistance heaters. When the molding conditions are on the border line of adding or removing heat from the circulating fluid, a coil attached to a cooling medium is inserted in the tank. A temperature sensing element activates the heating or cooling circuit for the temperature at which it is set. For temperatures above the boiling point of water nonaqueous fluids are used. It is essential to keep the fluid clean as rust scale and other contaminants seriously reduce the efficiency of the heat removal. When operated at high temperatures extreme care must be used in the selection and maintenance of the connecting hoses. A ruptured connector may result in serious burns (66).

Mold temperature control units will accurately control mold temperature. A unit attached to a mold running a half pound shot of general purpose styrene at 83 cycles/hr was instrumented. The inlet and outlet mold temperatures were read every 6 sec. They were charted with the heat on-off and water on-off controls of the unit, the cycle time of the machine and the mold temperature. In
a typical case with the cooling water and heating elements each cycling alternately every three shots, the inlet water temperature varied from 88.5 to 90.3°F, and the outlet water temperature from 89.7 to 91.5°F. The temperature difference between the outlet and inlet water was plotted for each 6-sec reading. It varied from 2.0 to 2.9°F with a mean of .9°F. The cycle of the curve followed that of the units heating cooling cycle. The mold temperature as read by a dial thermometer and pyrometer showed no readable change. By changing the molding conditions slightly so that cooling water was used all the time in the mold temperature unit the outlet temperature was 77.5°F and the inlet temperature varied between 73.4 and 73.6°F. The variation between the difference of the inlet and outlet water never exceeded 0.2°F. The dial thermometer and pyrometer in the mold showed no change. In the first instance 4590 Btu/hr were removed and in the latter 4740. These figures show that commercial units can produce accurate and consistent mold temperature control which is required for proper molding.

Heat Transfer

The three methods for exchanging heat are radiation, convection, and conduction. We are primarily concerned with convection and conduction. In the coolers for the molding machines the heat from the hot oil is exchanged into the tube walls, and from the tube walls into the circulating water. In the mold the heat from the plastic is transferred to the cavities and cores, which in turn transfers the heat to the mold temperature circulating medium. Some of the factors which affect the rate and amount of heat transfer are material of the container, size and shape of the container, rate of flow of both materials, temperature, viscosity, specific heat, thermal conductivity, density, and surface conditions of both sides of the container. The mathematics of these processes have not been quantitatively completed. Notwithstanding, a qualitative discussion of some of the factors affecting heat transfer is valuable.

The rate of heat removal equals the overall heat transfer coefficient times the area of exposed surface, times the difference in temperature between the two fluids (plastic and water).

\[
Q = UA \Delta t \tag{2-5}
\]

\[
Q = \text{rate of heat removal (Btu/hr)}
\]

\[
U = \text{overall heat transfer coefficient Btu/(hr) (ft}^2\) (°F)} \tag{2-6}
\]

\[
A = \text{area (ft}^2\)
\]

\[
\Delta t = \text{difference in temperature of the two fluids (°F).}
\]

This equation shows, as one would expect, that the lower the temperature of the cooling medium, the faster the heat removal. The rate could also be increased by increasing the material temperature. This would be self-defeating
because the higher removal rate would not compensate for the additional amount of heat to be removed, thus lengthening the cycle. The lower limit of the cooling temperature is the molding condition. Molds that are too cold may not fill, may develop surface blemishes and lower some physical properties.

The area of the cooling surface is limited by the geometry of the mold. Table 2-6 shows the effect of different size cooling channels. Using a 3/8-in. pipe instead of a 1/8-in. one will increase the cooling rate by a factor of 1.8. Large cooling channels are one of the easiest ways to reduce cycle time. Unfortunately this is often overlooked in mold design.

By use of electrical analogies the overall heat transfer coefficient is described as:

\[
\frac{1}{U} = \frac{1}{h_1} + \frac{1}{h_2} + \frac{1}{h_3} + \ldots + \frac{X}{k}
\]

\[X = \text{thickness of wall (ft)}\]
\[k = \text{thermal conductivity of wall} \quad \text{Btu}/(\text{hr})(\text{ft}^2) = \text{Btu}/(\text{hr})(\circ\text{F})(\text{ft})\]
\[h = \text{individual heat transfer coefficients Btu}/(\text{hr})(\text{ft}^2)(\circ\text{F})\]

This equation leads to some very interesting conclusions. It is important to notice that the heat transfer rate is controlled by the coefficient at the point of maximum resistance. For example ignoring \(X/k\), if there are only two coefficients, \(h_1 = 20\) and \(h_2 = 1000\), \(U\) would equal 19.6. Suppose \(h_2\) were changed from 1000 to 500, then \(U\) would equal 19.23. Therefore, even though one coefficient were changed by 50\% it would only change the total coefficient by 2\%. The film coefficient for water in the cooling system is approximately 1500. However, if scales, sludge and dirt enter the system this can drop to as low as 200 introducing serious resistance to heat transfer and probable increase in mold cycles. Therefore, clean circulating water and cooling channels are very important (67).

For molds the \(X/K\) factor is important. The rate of heat removal will vary directly with the thermal conductivity of the mold material. Therefore if the \(K\) for beryllium is 70 and steel 24 the beryllium will remove or add heat to the plastic approximately three times as fast as steel. This is an important factor in mold material selection (27). It is also obvious that the closer the cooling channel is to the plastic (a minimum \(X\)) the higher the rate of heat removal. Cooling channel location should be designed so that there is even cooling of the mold surface. Since heat removal varies directly with the distance between the cooling channel and the mold, equally spaced circles from the cooling channel will be roughly the same temperature. They can be drawn on a mold layout and a good indication of the temperature profile of the cavity or core obtained (68).

It is also evident that the highest heat transfer coefficient will occur when the
<table>
<thead>
<tr>
<th>Nominal Pipe Size</th>
<th>Tap Drill Used in Mold</th>
<th>Ratio of Cooling Area to Area of 1/8 in pipe</th>
<th>Surface ft² per ft of length</th>
<th>I.D. Circum. (in)</th>
<th>I.D. Area (in²)</th>
<th>Capacity at 1 Ft/sec</th>
<th>lb/hr</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8</td>
<td>5/16</td>
<td>1.0</td>
<td>0.0818</td>
<td>0.0767</td>
<td>0.3125</td>
<td>0.24</td>
<td>120</td>
</tr>
<tr>
<td></td>
<td>7/16</td>
<td>1.4</td>
<td>0.115</td>
<td>0.1503</td>
<td>0.4375</td>
<td>0.47</td>
<td>234</td>
</tr>
<tr>
<td></td>
<td>9/16</td>
<td>1.8</td>
<td>0.147</td>
<td>0.2485</td>
<td>0.5625</td>
<td>0.77</td>
<td>387</td>
</tr>
<tr>
<td></td>
<td>11/16</td>
<td>2.2</td>
<td>0.180</td>
<td>0.3712</td>
<td>0.6875</td>
<td>2.2</td>
<td>580</td>
</tr>
</tbody>
</table>
cooling channels are directly in the cavity or core. If put in the surrounding mold base the controlling coefficient will be between the cavity and the mold base. Interface losses are very significant and should be avoided if possible. This is one of the advantages of EDMing cavities in one block.

The ability of the plastic to change temperature is a factor in cooling the part. This is called the thermal diffusivity and is defined as the thermal conductivity divided by the product of the specific heat and the density. There is nothing the molder can do to change this as it is an inherent property of the material. It will explain why some materials cool more readily than others (69).

It stands to reason that the velocity of the cooling media would affect the heat transfer rate. From dimensional analysis and experimental work the heat transfer coefficient is affected by, among other things, the Reynolds number (p. 214). There is a velocity factor in the Reynolds number. When the number is below 2100 there is laminar flow and the heat transfer coefficient (inside a tube) varies as the 1/3 power of the velocity. Above 2100, turbulent flow, it varies as the 0.8 power of the velocity. The probable reason for this is that the turbulent flow provides better mixing and the metal-water interface is broken more often. In turbulent flow, for example, if water flowing through the tube had a film coefficient of heat transfer of 300 Btu/(h°F)(ft²)(°F) at a given velocity, and its velocity were doubled, the new film coefficient would be 300 (20.8) or 522. This means that increasing the velocity of the cooling fluid will increase the rate of heat transfer. This should not be overlooked. It may be necessary to increase the pumping capacity of the mold temperature control unit. The amount of fluid circulating can be easily determined with a water meter. With this information and the cooling channel dimensions the Reynolds number can be calculated.

When mold cooling seems inadequate the first things to be done are to clean the cooling system and mold channels, increase the velocity of the cooling medium and lower its temperature. If these methods do not work, consideration must be given to enlarging or adding to the cooling channels. Similarly, the heat exchanger for cooling the oil in the molding machine should be kept clean and periodically examined. If the machine overheats and the water temperature is normal, the cooler should be cleaned. If that does not help there probably is a malfunction in the hydraulic system permitting oil to bypass and generate heat.

The heat exchangers for cooling oil consists of tubes through which the cooling water flows, and a shell through which the hot oil, to be cooled, flows. They are mainly single pass exchanges; that is, the liquids flow in one direction. They can be connected in two ways. In parallel, the hot oil and the cold water enter at the same end so that the cooler oil and heated water will emerge at the other end. In counter flow the hot oil will enter at one end and the cold water will enter at the opposite end. Molding machines are connected counter flow,
since it will remove approximately 10% more heat from the oil.

This has relevance in mold cooling, as a mold is a heat exchanger with the hot plastic as a heat source and water as the cooling medium. Most molds have the hottest section at the sprue, primarily because of the radiation effect of the outside of the mold base. Attaching cold water to the outside of the mold (analogous to counter flow) will remove more heat than putting the cooling water directly into the sprue section first. Properly designed molds will permit the plastics engineer to adjust mold temperature accordingly.

**Mold Temperature Control**

The two reasons for providing for good mold temperature control are (a) economic and (b) part quality. The temperature control system includes the cooling fluid, means for its circulation, method of temperature control, and cooling channels in the mold. Its purpose is to remove heat from the plastic part at a controlled rate. The goal is the removal of heat as rapidly as possible so that the part can be removed from the mold in a condition which will result in acceptable pieces. This cannot be done without a good temperature control system which will permit the molding conditions to establish the mold temperature rather than the adequacy of the equipment.

An incorrect and inconsistent mold temperature will create serious difficulties. We shall assume an adequate system and discuss some of the problems caused by incorrect mold temperature. It is not always possible or necessary to predict the best temperature for a given mold and material. With thermostatically controlled temperatures, trial and error is not difficult. In many instances there will be several different temperatures maintained for different parts of the mold.

To cool any given part a specific number of BTU will have to be removed. Equation 2-5 shows that the greater the temperature difference between the plastic and the cooling fluid the higher the heat removal rate. Therefore, a lower mold temperature will permit the part to be removed more quickly.

**Material or Mold Temperature Too High.** If the temperature is too high, cycle time increases. This is not the only disadvantage. Some of the others are described below.

Because the plastic will remain fluid longer with higher temperatures, there is a greater tendency for the material to flash. The gate will remain open longer permitting more material to be packed into the cavity. This excessive packing particularly at the gate may lead to an over stressed part and difficulties in ejection. The packed material either deforms the steel and/or adheres to it more strongly than usual, increasing the possibility of sticking.

Since the part will probably be softer on ejection there is a greater chance for the ejectors to force their way into the material. Furthermore, an overheated
mold may (a) not allow the sprue to solidify resulting in it sticking, and (b) cause excessive shrinkages, sink marks, and burning, although these are less frequent. If one side of the mold overheats the thermal expansion of the mold may cause one side or plate to seize. This can pull the mold off the platen.

When hot thermoplastic hits a cold mold, part of the polymer freezes against the wall and some is stretched or “oriented” in the direction of flow. This orientation is greatest near the surface of the mold and decreases towards the center of the part. The amount will depend in some measure upon the mold temperature. It can be helpful or troublesome. Orientation leads to molded-in stress, and highly oriented parts which are usually not desirable. The part has a higher tensile strength in the direction of flow because it has more carbon-carbon linkages than perpendicular to flow, where the main cohesive forces are the weaker electrostatic bonds. This is discussed in detail in Chapter 3.

**Molds Too Cold.** Molds that are too cold will also cause considerable difficulty. If the mold is filled the plastic most distant from the gate will have a lot less material than that close to the gate, which will be warm enough to receive some packing during injection. This will leave an uneven density distribution causing severe molded-in stresses.

Moreover, the material might freeze before it fills the cavity giving incomplete shots. There may not be enough material forced into the mold before the material stops flowing. This can be caused either by premature freezing of the plastic in the mold or premature freezing of the gate. In these instances there will not be enough material in the part, which can mean voids, excessive shrinkage, and severe reduction of the mechanical properties of the part, the insufficiency of material will emphasize sink marks.

The cold mold surface may also cause tails, tears, and surface smears which are caused by the skidding of cold material that does not remelt into the polymer, as well as emphasizing the negative aspect of weld lines, which are the junction of two fronts of molten polymer.

A characteristic of cold molds are tiny ripples on the surface, which are in the form of wavefront perpendicular to the direction of flow.

These are just some of the major problems caused by incorrect mold temperature setting. These are all compounded if the cooling system lacks the capacity or instrumentation to maintain a consistent temperature. Varying mold temperatures are even worse than incorrect temperatures and make quality molding impossible.

**Mold Cooling Channels.** Before discussing mold cooling channels, we reemphasize that materials for cavities and cores must be evaluated in terms of their thermal properties. This concept extends even to the use of a combination of metals. It might be desirable, for example, to hollow out the core of a tumbler mold and put in a beryllium insert to improve its thermal conductivity.
In evaluating the economics of a mold, it is rare that extra money spent for cooling is not quickly recouped. Also, the cooling system (excluding the mold) must have enough cooling capacity and pumps able to deliver water at high velocities. Dropping the mold temperature and increasing the velocity of the water to at least Reynolds 3000 will increase the rate of heat removal, and the amount of heat removed from the mold. Attention should be paid to the way the cooling water is attached to the mold cooling channels. For example, if there are a number of cooling channels drilled parallel, they should be hooked up in parallel from the water supply and not in series. The outlets should be at least as large as the inlets.

Figure 2-28 shows an adequate cooling channel pattern for a mold plate. The pattern on the top is inferior because the left-hand side of the plate will be at a lower temperate than the right-hand side. Depending on what is molded, a significant difference in the plastic part might be observed. The lower design shows the same holes but with a different baffle arrangement. The cooling could be attached either in parallel or counter flow. By rearranging the baffles and plugs any type of local mold temperature control could be achieved.

Figure 2-29 shows what can be done to increase the cooling of a core. The core is hollowed out. A stainless steel insert is turned to the same taper with a spiral groove running from top to bottom. The cooling fluid enters from the bottom, fills up the spiral, and drops down a hole drilled in the center to the outlet. This will give tremendously superior temperature control when compared to conventional bubblers. While this, too, is more expensive to build it is the most economical mold design.

Cores, cavities, and pins are sometimes cooled with bubblers. Figure 2-30 shows the series cooling of a pin. The water enters the first pin, flows over a blade into the second pin, and so on. This is a very poor design. The first pin would be much much cooler than the last. Even a slight bit of corrosion or dirt can clog or seriously restrict the flow. Because of the high resistance there will be a minimum velocity. The proper way for such cooling is shown in Figure 2-31. Two holes are drilled in the plate, one under the other. The metal between them is tapped and a bubbler pipe of a noncorrosive metal is screwed in. The water flow is equal in all channels. The clogging of one pin will not effect the others. Maximum velocity is obtainable. The "in" channel must be directed to the top channel so that the tube will fill up with water before it overflows. If the water entered the outside and the in-channel had a larger capacity the pin might never fill with water. The references on mold cooling contain many ingenious ways of increasing the cooling capacity of molds (70-75A).
Figure 2-28  Cooling channels: (a) adequate design, and (b) preferred design.

Automation

Many times one hears the expression, “automatic machines” when referring to automatic molding. This is a misnomer; all machines today are automatic. What makes automatic molding automatic is the mold. There are a number of requirements for automatic molding:
1. The machine must be capable of consistent, repetitive action.

2. The mold must clear itself automatically. This means that all the parts have to be ejected using a runnerless mold, or that the gate and parts have to be ejected in a conventional manner and fall free of the mold. There usually is some method for assisting in the removal of the pieces and gates, in the form of a wiper mechanism or an air blast. Some systems weigh the shot after ejection and stop the machine or sound an alarm if the shot is too light.

3. Indicating that a part is stuck is necessary. All machines used automatically must have a low pressure closing system which prevents the machine from closing under full pressure if there is any obstruction between the dies. The
machine is shut off and/or an alarm is sounded.

Automatic molding, which usually produces better parts more rapidly, does not necessarily eliminate the operator. Many times an operator is present to pack the parts and perform secondary operations. However, some new systems automate this function as well. Usually in automatic molding an experienced person attends several machines. Unless the powder feed and part removal are automated he will take care of them.

Automation means replacing high labor costs with high capital costs for molds and parts handling equipment. Excellent machinery, good molds, trained employees and managerial skill are all required. When the quantity of a part permits, it is a very satisfactory and economical operation (79, 80, and 81).

**Mold Maintenance**

Mold maintenance can be done either on an emergency basis or between runs. Management policy will determine whether a mold should be fixed during the run. It will depend on how badly the customer needs the part, the length of time for a temporary repair, the time for a permanent repair, the availability of