PRODUCT DESIGN
for MANUFACTURE
and ASSEMBLY

Geoffrey Boothroyd

Peter Dewhurst

Winston Knight

University of Rhode Island
Kingston, Rhode Island

8

Design for Injection Molding

8.1 INTRODUCTION

Injection molding technology is a method of processing predominantly used for thermoplastic polymers. It consists of heating thermoplastic material until it melts, then forcing this melted plastic into a steel mold, where it cools and solidifies. The increasingly sophisticated use of injection molding is one of the principal tools in the battle to produce elegant product structures with reduced part counts. Perhaps the most widely recognized, innovative new product design in this context is the Proprinter developed by IBM as the domestic competitor to the Japanese dot-matrix printers. Plastic components in the Proprinter incorporate the functions of cantilever springs, bearings, support brackets and fasteners into single snap-fit components. The result of this integration of features into single complex parts is a reduction of part count from 152 to 32, with a corresponding reduction of assembly time from 30 to 3 min [1], when compared with the Epson printer which IBM had previously been remarketing from Japan.

In order to exploit the versatility of injection molding technology for economical manufacture, it is necessary to understand the basic mechanisms of the process and related aspects of the molding equipment and materials used. Also, since injection molding is a process which utilizes expensive tooling and equipment, it is vital to be able to obtain part and tooling cost estimates at the earliest stages of design. Only in this way can the design team be sure that the choice of the process is a correct one and that maximum economic advantage will be obtained from the process. For these reasons, this chapter will first present a review of injection molding materials and the injection molding process. This will be followed by the description of a procedure for estimating the cost of injection molded parts, which is applicable to the early phase of product design.

8.2 INJECTION MOLDING MATERIALS

It is not possible to injection-mold all polymers. Some polymers like PTFE (poly tetra-fluro ethylene), cannot be made to flow freely enough to make them suitable for injection molding. Other polymers, such as a mixture of resin and glass fiber in woven or mat form, are unsuitable by their physical nature for use in the process. In general, polymers which are capable of being brought to a state of fluidity can be injection-molded.

The vast majority of injection molding is applied to thermoplastic polymers. This class of materials consists of polymers which always remain capable of being softened by heat and of hardening on cooling, even after repeated cycling. This is because the long-chain molecules of the material always remain as separate entities and do not form chemical bonds to one another. An analogy can be made to a block of ice that can be softened (i.e., turned back to liquid), poured into any shape cavity, then cooled to become a solid again. This property differentiates thermoplastic materials from thermosetting ones. In the latter type of polymer, chemical bonds are formed between the separate molecule chains during processing. In this case the chemical bonding, referred to as cross linking, is the hardening mechanism. Thermosetting polymers are generally more expensive to mold than thermoplastics and represent only about five percent of plastics processing. In this chapter we will concentrate exclusively on injection molding of thermoplastics.

In general, most of the thermoplastic materials offer high impact strength, good corrosion resistance, and easy processing with good flow characteristics for molding complex designs. Thermoplastics are generally divided into two classes: namely, crystalline and amorphous. Crystalline polymers have an ordered molecular arrangement, with a sharp melting point. Due to the ordered arrangement of molecules, the crystalline polymers reflect most incident light and generally appear opaque. They also undergo a high shrinkage or reduction in volume during solidification. Crystalline polymers usually are more resistant to organic solvents and have good fatigue and wear-resistant properties. Crystalline polymers also generally are denser and have better mechanical properties than amorphous polymers. The main exception to this rule is polycarbonate, which is the amorphous polymer of choice for high-quality transparent moldings, and has excellent mechanical properties.

The mechanical properties of thermoplastics, while substantially lower than those of metals, can be enhanced for some applications through the addition of glass fiber reinforcement. This takes the form of short-chopped fibers, a few millimeters in length, which are randomly mixed with the thermoplastic resin.

The fibers can occupy up to one third of the material volume to considerably improve the material strength and stiffness. The negative effect of this reinforcement is usually a decrease in impact strength and an increase in abrasiveness. The latter also has an effect on processing since the life of the mold cavity is typically reduced from about 1,000,000 parts for plain resin parts to about 300,000 for glass-filled parts.

Perhaps the main weakness of injection-molded parts is the relatively low service temperatures to which they can be subjected. Thermoplastic components can only rarely be operated continuously above 250°C, with an absolute upper service temperature of about 400°C. The temperature at which a thermoplastic can be operated under load can be defined qualitatively by the heat deflection temperature. This is the temperature at which a simply supported beam specimen of the material, with a centrally applied load, reaches a predefined deflection. The temperature value obviously depends upon the conditions of the test and the allowed deflection and for this reason, the test values are only really useful for comparing different polymers.

Table 8.1 lists the more commonly molded thermoplastics together with typical mechanical property values.

8.3 THE MOLDING CYCLE

The injection molding process cycle for thermoplastics consists of three major stages as shown in Fig. 8.1: (1) injection or filling, (2) cooling, and (3) ejection and resetting. During the first stage of the process cycle, the material in the molten state is a highly nonlinear viscous fluid. It flows through the complex mold passages and is subject to rapid cooling from the mold wall on the one hand and internal shear-heating on the other. The polymer melt then undergoes solidification under the high packing and holding pressure of the injection system. Finally the mold is opened, the part is ejected and the machine is reset for the next cycle to begin.

8.3.1 Injection or Filling Stage

The injection stage consists of the forward stroke of the plunger or screw injection unit to facilitate flow of molten material from the heating cylinder through the nozzle and into the mold. The amount of material to be transferred into the mold is referred as the shot. The injection stage is accompanied by a gradual increase in pressure. As soon as the cavity is filled, the pressure increases rapidly, and packing occurs. During the packing part of the injection stage, flow of material continues, at a slower rate, to account for any loss in volume of the material due to partial solidification and associated shrinkage. The packing time depends on the properties of the materials being molded. After packing, the injection plunger is withdrawn or the screw is retracted and the

Table 8.1 Commonly Used Polymers in Injection Molding

Thermoplastic	Yield strength (MN/m²)	Elastic modulus (MN/m²)	Heat deflection temperature (° C)	Cost ($/kg)
High-density polyethylene	23	925	42	0.90
High-impact polystyrene	20	1,900	77	1.12
Acrylonitrile-butadiene-styrene(ABS)	41	2,100	99	2.93
Acetal (homopolymer)	66	2,800	115	3.01
Polyamide (6/6 nylon)	70	2,800	93	4.00
Polycarbonate	64	2,300	130	4.36
Polycarbonate (30% glass)	90	5,500	143	5.54
Modified polyphenylene oxide (PPO)	58	2,200	123	2.75
Modified PPO (30% glass)	58	3,800	134	4.84
Polypropylene (40% talc)	32	3,300	88	1.17
Polyester teraphthalate (30% glass)	158	11,000	227	3.74

pressure in the mold cavity begins to drop. At this stage, the next charge of material is fed into the heating cylinder in preparation for the next shot.

8.3.2 Cooling or Freezing Stage

Cooling starts from the first rapid filling of the cavity and continues during packing and then following the withdrawal of the plunger or screw with the resulting removal of pressure from the mold and nozzle area. At the point of pressure removal, the restriction between the mold cavity and the channel conveying material to the cavity, referred to as the gate of the mold, may still be relatively fluid, especially on thick parts with large gates. Because of the pressure drop, there is a chance for reverse flow of the material from the mold until the material adjacent to the gate solidifies and the sealing point is reached. Reverse flow is minimized by proper design of the gates such that quicker sealing action takes place upon plunger withdrawal [2,3].

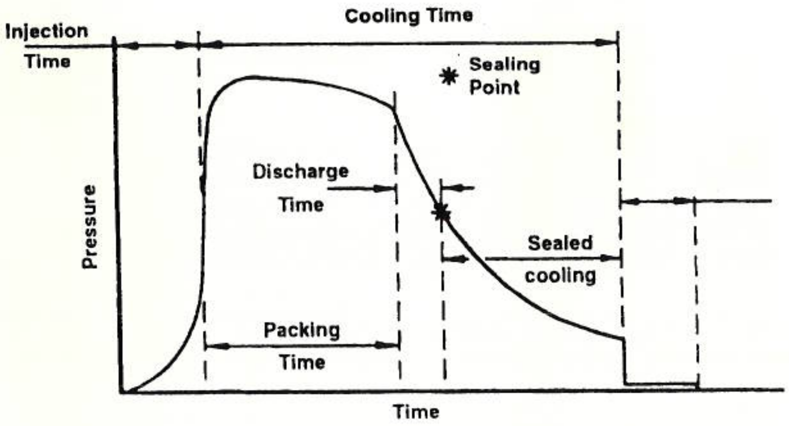

Figure 8.1 Injection molding cycle.

Following the sealing point, there is a continuous drop in pressure as the material in the cavity continues to cool and solidifies in readiness for ejection. The length of the sealed cooling stage is a function of the wall thickness of the part, the material used and the mold temperature. Because of the low thermal conductivity of polymers, the cooling time is usually the longest period in the molding cycle.

8.3.3 Ejection and Resetting Stage

During this stage, the mold is opened, the part is ejected, and the mold is then closed again in readiness for the next cycle to begin. Considerable amounts of power are required to move the often massively built molds, and mold opening and part ejection are usually executed by hydraulic or mechanical devices. Although it is economical to have quick opening and closing of the mold, rapid movements may cause undue strain on the equipment, and if the mold faces come into contact at speed, this can damage the edges of the cavities. Also, adequate time must be allowed for the mold ejection. This time depends on the part dimensions which determine the time taken for the part to fall free of moving parts between the machine platens. For parts to be molded with metal inserts, resetting involves the reloading of inserts into the mold. After resetting, the mold is closed and locked thus completing one cycle.

8.4 INJECTION MOLDING SYSTEMS

An injection molding system consists of the machine and mold for converting, processing and forming of raw thermoplastic material, usually in the form of pellets, into a part of desired shape and configuration. Figure 8.2 shows a schematic view of a typical injection molding system. The major components of an injection molding system are: the injection unit, the clamp unit, and the mold. These are described briefly in the following sections.

8.4.1 Injection Unit

The injection unit has two functions: to melt the pellets or powder, and to inject the melt into a mold. The most widely used types of injection units, are (i) conventional units consisting of a cylinder and a plunger which forces the molten plastic into the mold cavity, and (ii) reciprocating screw units, consisting of a barrel or cylinder and a screw which rotates to melt and pump the plastic mix from the hopper to the end of the screw and then moves forward to push the melt into the mold.

Of the two types, reciprocating screw injection units are considered to be of better design because of their improved mixing action. The motion of the polymer melt along the screw flights helps to maintain a uniform melt temperature. It also facilitates better blending of the materials and any coloring agents resulting in the delivery of more uniform melt to the mold. Because of these advantages, reciprocating screw units are found on the majority of modern injection molding machines.

The injection units are usually rated with two numbers: the first is the shot capacity defined as the maximum volume of polymer that can be displaced by one forward stroke of the injection plunger or screw. The shot capacity for other materials can be calculated by using the ratio of specific gravities. It is usually recommended that an injection unit be selected so that the required shot sizes will fall within 20 to 80% of the rated capacity.

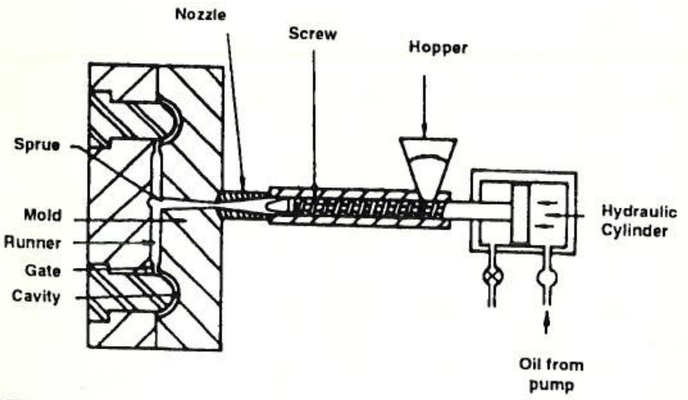

Figure 8.2 Injection molding system.

The second rating number is the plasticating rate, which is the amount of material which can be plasticized or softened into a molten form by heating in the cylinder of the machine in a given time. This number is usually expressed as the number of pounds of polystyrene material that the equipment can heat to molding temperature in one hour. For further information on polymer processing requirements consult references [2–6].

8.4.2 Clamp Unit

The clamp unit has three functions: to open and close the mold halves, to eject the part and to hold the mold closed with sufficient force to resist the melt pressure inside the mold as it is filled. The required holding force typically varies between 30 and 70 MN/m^2 of projected area of the part (approximately 2 to 5 $tonF/in^2$). The pressure developed in the mold during filling and packing, and the shrinkage of the part onto its cores, may cause the part to stick, thus making the separation of the two mold halves difficult. The magnitude of the initial opening force required depends on packing pressure, material and part geometry (depth and draft) and is approximately equal to 10 to 20% of the nominal clamp force [3,7].

There are two common types of clamp designs:

(i) Linkage or toggle clamp—This design utilizes the mechanical advantage of a linkage to develop the force required to hold the mold during the injection of the material. Mechanical toggle clamps have very fast closing and opening actions and are lower in cost than alternative systems. The major disadvantage is that the clamp force is not precisely controlled and for this reason they toggle clamps to be found only on smaller machines.

(ii) Hydraulic clamp units—These use hydraulic pressure to open and close the clamp, and to develop the force required to hold the mold closed during the injection phase of the cycle. The advantages of this type of design are long-term reliability and precise control of clamp force. The disadvantage is that hydraulic systems are relatively slow and expensive compared to toggle clamp systems.

After the mold halves have opened, the part which has a tendency to shrink and stick to the core of the mold (usually the half of the mold furthest from the injection unit) has to be ejected by means of an ejector system provided in the clamping unit. The force required to eject the part is a function of material, part geometry and packing pressure and is usually less than 1% of the nominal clamp force [8].

8.5 INJECTION MOLDS

Molds for injection molding are as varied in design, degree of complexity and size as are the parts produced from them. The functions of a mold for thermoplastics are basically to impart the desired shape to the plasticized polymer and then to cool the molded part.

A mold is made up of two sets of components: (1) the cavities and cores, and (2) the base in which the cavities and cores are mounted. The size and weight of the molded parts limit the number of cavities in the mold and also determine the equipment capacity required. From consideration of the molding process, a mold has to be designed to safely absorb the forces of clamping, injection and ejection. Also, the design of the gates and runners must allow for efficient flow and uniform filling of the mold cavities.

8.5.1 Mold Construction and Operation

Figure 8.3 illustrates the parts in a typical injection mold. The mold basically consists of two parts: a stationary half (cavity plate), on the side where molten polymer is injected, and a moving half (core plate) on the closing or ejector side of the injection molding equipment. The separating line between the two mold halves is called the parting line. The injected material is transferred through a central feed channel, called the sprue. The sprue is located on the sprue bushing and is tapered to facilitate release of the sprue material from the mold during mold opening. In multicavity molds, the sprue feeds the polymer melt to a runner system, which leads into each mold cavity through a gate.

The core plate holds the main core. The purpose of the main core is to establish the inside configuration of the part. The core plate has a backup or support plate, The support plate in turn is supported by pillars against the U-shaped structure known as the ejector housing, which consists of the rear clamping plate and spacer blocks. This U-shaped structure which is bolted to the core plate provides the space for the ejection stroke also known as the stripper stroke. During solidification the part shrinks around the main core so that when the mold opens, part and sprue are carried along with the moving mold half. Subsequently, the central ejector is activated, causing the ejector plates to move forward so that the ejector pins can push the part off the core.

Both mold halves are provided with cooling channels through which cooled water is circulated to absorb the heat delivered to the mold by the hot thermoplastic polymer melt. The mold cavities also incorporate fine vents (0.02 to 0.08 mm by 5 mm) to ensure that no air is trapped during filling.

8.5.2 Mold Types

The most common types of molds used in industry today are: (1) two-plate molds; (2) three-plate molds; (3) side-action molds and (4) unscrewing molds.

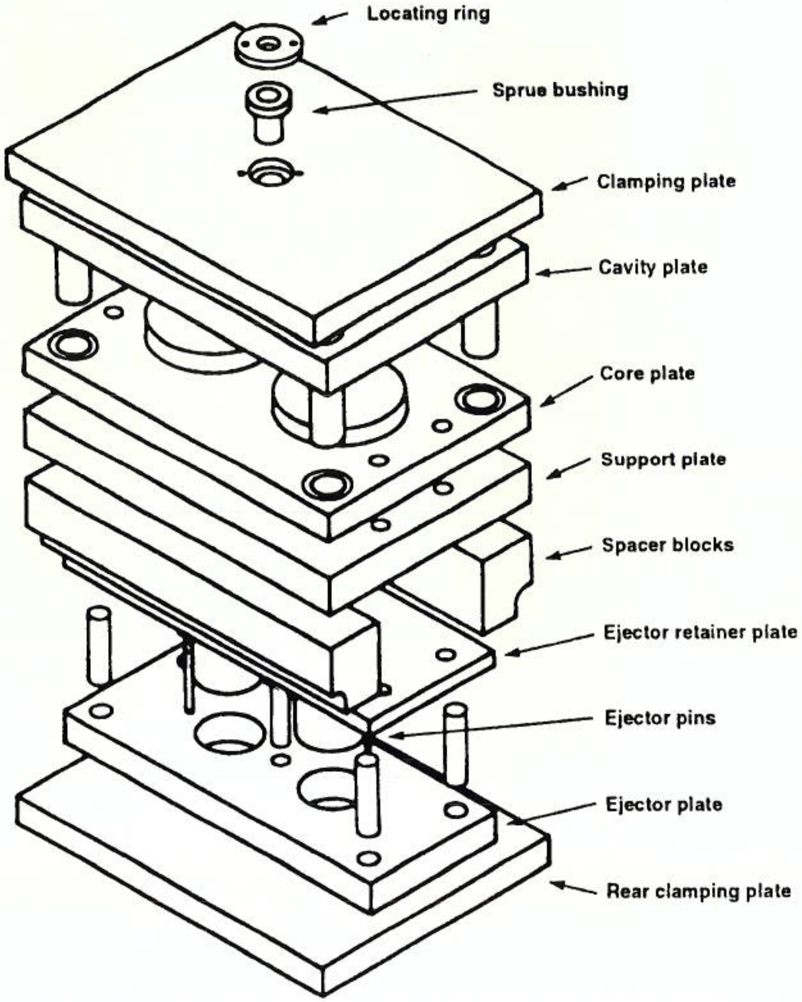

Figure 8.3 Mold.

A two-plate mold consists of two active plates (cavity and core plates in Fig. 8.3) into which the cavity and core inserts are mounted as shown in Fig. 8.4. In this mold type, the runner system, sprue, runners and gates solidify with the part being molded and are ejected as a single connected item. Thus the operation of a two-plate mold usually requires continuous machine attendance. The machine operator must spend time separating the runner system from parts

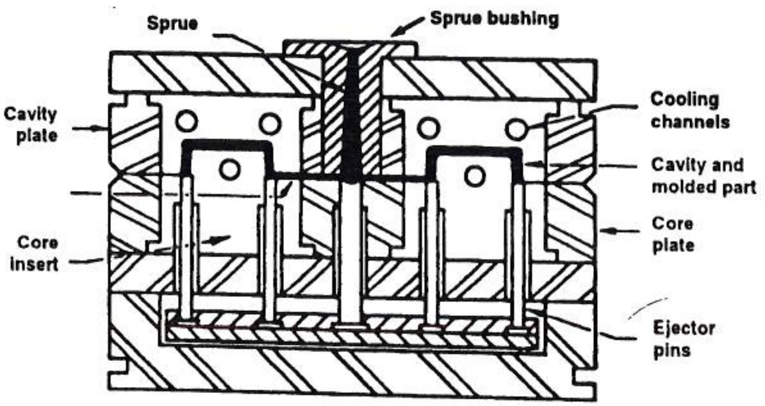

Figure 8.4 Two-plate injection mold.

(which break easily at the narrow gates) and periodically cutting the runner systems into small pieces which can be reintroduced into the machine hopper. This task is accomplished in an ancillary piece of equipment which operates like a small wood chipper and the chips are somewhat confusingly referred to as regrind.

The three-plate mold consists of: (i) the stationary or runner plate, which contains the sprue and half of the runner; (ii) the middle or cavity plate, which contains the other half of the runner, the gates and cavities and is allowed to float when the mold is open; and (iii) the movable or core plate, which contains the cores and the ejector system. This type of mold design facilitates separation of the runner system and the part when the mold opens; the two items usually fall into separate bins below the mold.

For very high rates of production, full automation can be achieved with a hot runner system, sometimes called a runnerless molding system. In this system, which also uses three main plates, the runner is contained completely in the fixed plate which is heated and insulated from the rest of the cooled mold. The runner section of the mold is not opened during the molding cycle. The advantages of this design are that there are no side products (gates, runner, or sprues) to be disposed of or reused, and there is no need for separation of the gate from the part. Also, it is possible to maintain a more uniform melt temperature.

Side-acting molds are used in molding components with external depressions or holes parallel to the parting plane. These features are sometimes referred to as undercuts or cross features. These undercuts prevent molded parts being removed from the cavity in the axial direction and are said to create a die-locked situation. The usual way of providing the side-action needed to release the part is with side cores mounted on slides. These are activated by angle pins, or by air or hydraulic cylinders which pull the side cores outward during opening of the mold. Because of this action, the side core mechanisms are often referred to as side-pulls.

The mechanism of an angle-pin side-action mold for a part with an undercut formed by a hole is illustrated in Fig. 8.5. The slide which carries the secondary side core pin is moved by the angle pin mounted in the stationary half of the mold. As the two halves of the mold move apart during mold opening, the slide, which is mounted on the moving plate, is forced to move sideways by the angle of the pin. This allows the undercut to become free of the core pin and the part can then be ejected. Note that one side-pull is needed for each cross-feature or group of cross-features which lie on a particular axis. Molds have

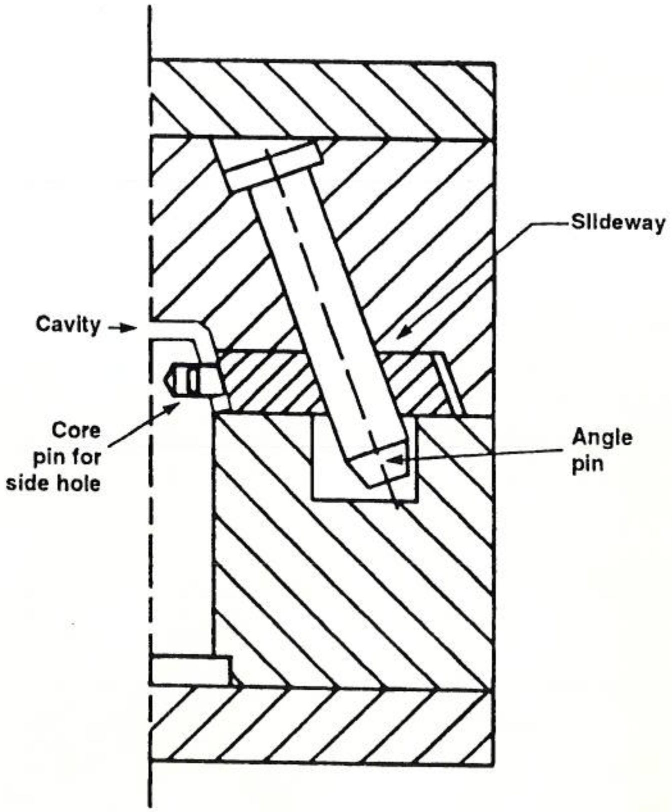

Figure 8.5 Angle-pin activated side-pull.

been built with as many as nine separate side-pulls to release particularly complex parts.

The mechanism for unscrewing molds is shown in Fig. 8.6. The rack and pinion gear mechanism shown in the illustration is the most common method used to free the undercuts formed by internal or external threads. With this method, a gear rack moved by a hydraulic cylinder engages with a spur gear which is attached to the threaded core pin. The rotating action imparted to the core pin through the gear transmission thus frees the undercuts formed by the threads. This additional unscrewing mechanism increases the mold cost and mold maintenance cost to a great extent, but eliminates the need for a separate thread-cutting operation. Note that external thread forms with axes which lie on the molding plane can be separated from the mold without the need for an unscrewing device.

A final category of mold mechanisms is required to mold depressions or undercuts on the inside of plastic parts. The design of a part with internal die-locking features of this type requires the mold maker to build the core pin retraction device within the main core. This is clearly much more difficult and expensive to manufacture than a corresponding side-pull on the outside of the cavity. In the latter case, adequate plate area can readily be provided for the machining of slideways. For this reason, the need for internal core retraction mechanisms, called lifters by mold makers, should be avoided whenever possible. The obvious way to do this is to replace internal depressions with either external ones or through holes.

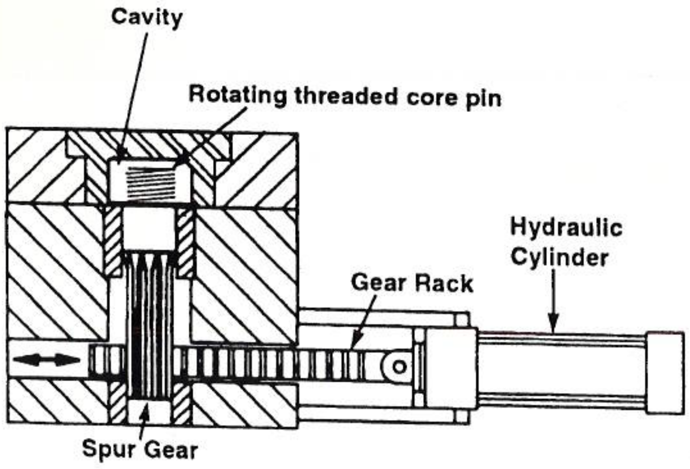

Figure 8.6 Unscrewing mold.

The ability of the injection molding process to handle unusual shapes is made possible by the fact that many different types of gates, ejection systems and mold movement mechanisms can be combined into one mold. The resulting mold may be highly complex and often extremely expensive to both build and maintain. However, it should be realized that the general rule for efficient manufacture is to incorporate as many features as possible into a single molded part, provided, of course, that the required number of parts to be produced is sufficiently large to justify the tool costs.

8.5.3 Sprue, Runner and Gates

A complete runner system with sprue and gates is shown in Fig. 8.4. The sprue bushing acts as an inlet channel for molten material from the heating chamber into the mold or runner system. The solid material in the form of a carrot called the "sprue" acts as a transition from the hot molten thermoplastic in the chamber to the relatively cooler mold. Runner systems are used in multiple cavity molds and act as channels to connect the sprue bush to the cavity gates. The gate, a constriction between the feed system and the mold cavity, serves several purposes. It freezes rapidly and prevents material from flowing out of the cavity when the injection pressure is removed. It provides an easy way of separating moldings from the runner system. It also suddenly increases the rate at which the polymer is sheared which helps to align the polymer chains for more effective cavity filling.

In multiple-cavity molds, great care is taken to balance the runner system in order to produce identical parts. Different-length runners of the same cross-sectional area would result in different cavity pressures with resulting size and density variations among the multicavity parts.

8.6 MOLDING MACHINE SIZE

Determination of the appropriate size of an injection molding machine is based primarily on the required clamp force. This in turn depends upon the projected area of the cavities in the mold and the maximum pressure in the mold during mold filling. The former parameter is the projected area of the part, or parts if a multicavity mold is used, and runner system, when viewed in the direction of mold opening, i.e., the area projected onto the surface of the mold cavity plate. The value for this parameter should not include any through holes molded in the direction of mold opening. Thus for a 15 cm diameter plain disk, the projected area is 176.7 cm^2. However, if the disk has a single 10 cm diameter through hole in any position, the projected area is 98.2 cm^2 since this is the area over which the polymer pressure will act during filling.

The size of the runner system depends upon the size of the part. Typical runner volumes as a percentage of part volume are shown in Table 8.2. As a

Table 8.2 Runner Volumes (Du Pont)

Part volume (cm^3)	Shot size (cm^3)	Runner %
16	22	37
32	41	27
64	76	19
128	146	14
256	282	10
512	548	7
1024	1075	5

first approximation, these figures will also be applied to give the projected area of the runner system as a percentage of the projected area of the part. Note, however, this is only strictly correct if a part is flat and if the runner system is the same thickness as the part.

Estimation of polymer pressure in the cavity during mold filling is a much more difficult problem. The flow characteristics of polymers are highly nonlinear, and mathematical models for mold filling can only be analyzed for individual runner and cavity geometries through the use of computer intensive numerical procedures. However, it appears that as a general rule, approximately 50 percent of the pressure generated in the machine injection unit is lost due to the flow resistance in the sprue, runner systems and gates [9]. This rule will be applied extensively in the costing analyses later in this chapter.

Example:

A batch of 15 cm diameter disks with a thickness of 4 mm are to be molded from ABS in a six-cavity mold. Determine the appropriate machine size:

(i) The projected area of each part equals 177 cm^2. From Table 8.2 the percentage increase in area due to the runner system is approximately 15 percent. Thus the total projected shot area will be

$$6 \times 1.15 \times 177 = 1221.3\ cm^2$$

(ii) The recommended injection pressure for ABS from Table 8.5 is 1000 bars. Thus the maximum cavity pressure is likely to be 500 bars or $500 \times 10^5\ N/m^2$.

(iii) The estimate of maximum separating force F is thus given by

$$F = (1221.3 \times 10^{-4}) \times 500 \times 10^5\ N = 6{,}106.5\ kN$$

Thus, if the available machines are those listed in Table 8.4, then the appropriate machine would be the one with a maximum clamp force of 8,500 kN.

This machine must be checked to ensure that it has a sufficient shot size and a large enough clamp stroke. The required shot size is the volume of the six disks plus the volume of the runner system. This equals

$$6 \times 1.15 \times (177 \times 0.4) = 489\ cm^3,$$

which can be seen to be easily within the maximum machine shot size of 3,636 cm^3.

The final check on machine suitability is the available machine clamp stroke. For the 8,500 kN machine this is shown to be 85 cm. This stroke is sufficient to mold a hollow part up to a depth of approximately 40 cm. For such a part, the 85 cm stroke would separate the molded part from both the cavity and the core with a clearance of approximately 5 cm for the part to fall between the end of the core and the cavity plate. This stroke is thus excessive for the molding of 4 mm thick flat disks. The stroke is, however, adjustable, and to speed up the machine cycle it would be reduced in this case to just a few centimeters.

8.7 MOLDING CYCLE TIME

After the appropriate machine size for a particular molded part s been established, the molding cycle time can next be estimated. This estimation is essential in any consideration of the merits of alternative part designs or the choice of alternative polymers. As described earlier in this chapter, the molding cycle can be effectively divided into three separate segments: namely, injection or filling time, cooling time and mold resetting time. Time estimates for these three separate segments will be established in this section.

8.7.1 Injection Time

A precise estimate of injection time requires an extremely difficult analysis of the polymer flow as it travels through the runners, gates, and cavity passages. This type of analysis would clearly not be justified as a basis for initial comparisons of alternative part design concepts. At this stage of design the position and number of gates and the size of the runner system would not be known. To circumvent this problem some major simplifying assumptions will be made about the machine performance and the polymer flow. First, modern injection molding machines are equipped with powerful injection units specifically to achieve the required flow rates for effective mold filling. It will be assumed that, at the commencement of filling, the full power of the injection unit is utilized and that the polymer pressure at the nozzle of the injector is that recommended by the polymer supplier. Under these circumstances, which may not be realizable for a particular mold design, the flow rate, using elementary mechanics, would be given by

$$Q = P_j/p_j\ m^3/s \tag{8.1}$$

where

P_j = injection power, W

and

p_j = recommended injection pressure, N/m^2

In practice the initial flow rate will gradually decrease as the mold is filled, due to both flow resistance in the mold channels and a constriction of the channels as the polymer solidifies against the walls. It will further be assumed that the flow rate suffers a constant deceleration to reach an insignificantly low value at the point at which the mold is nominally filled. Under these circumstances, the average flow rate would be given by

$$Q_{av} = 0.5\, P_j/p_j \ \mathrm{m^3/s} \qquad (8.2)$$

and the fill time would be estimated as

$$t_f = 2V_s p_j/P_j \ \mathrm{s} \qquad (8.3)$$

where

V_s = required shot size, m^3

Example:

For the 15 cm diameter disks molded in a 6-cavity mold, described in Sec. 8.6, the required shot size is 489 cm^3. The recommended injection pressure for ABS is 1000 bars or 100 MN/m^2. The available power at the injection unit of the 8,500 kN machine is 90 kW.
Thus the estimated fill time is

$$t_f = 2 \times (489 \times 10^{-6}) \times (100 \times 106)/(90 \times 10^3)$$
$$= 1.09\ \mathrm{s}$$

8.7.2 Cooling Time

In the calculation of cooling time, it is assumed that cooling in the mold takes place almost entirely by heat conduction. Negligible heat is transferred by convection since the melt is highly viscous and it is clear that radiation cannot contribute to the heat loss in a totally enclosed mold.

An estimation of cooling time can be made by considering the cooling of a polymer melt of initial uniform temperature T_i, between two metal plates, distance h apart, and held at constant temperature T_m. This situation is analogous to cooling of the wall of an injection-molded component between the mold cavity and core. The variation of temperature across the wall thickness and with changing time is described by the one-dimensional heat conduction equation:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \qquad (8.4)$$

Where

x = coordinate distance from center plane of wall normal to the plate surface, mm

T = temperature, °C

t = time, s

α = thermal diffusivity coefficient, mm^2/s

The thermal conductivity of thermoplastic materials is about three orders of magnitude lower than that of the steel mold. Under this situation it is reasonable to neglect the thermal resistance of the mold, which then merely becomes a heat sink at assumed constant temperature T_m. A classical series solution to this boundary condition applied to Eq. (8.4) has been given by Carslaw and Jaeger [10].

Ballman and Shusman [11] suggested an estimate of the cooling time based on truncating the Carslaw and Jaeger general series solution to just the first term. Mold opening and ejection are assumed to be permissible when the injected polymer has cooled to the point where the highest temperature in the mold (at the thickest wall center plane) equals T_x, the recommended ejection temperature. With this assumption the first-term solution for the cooling time is given by

$$t_c = \frac{h^2 \mathrm{max}}{\pi^2 \alpha} \log_e \frac{4(T_i - T_m)}{\pi(T_x - T_m)} \ \mathrm{s} \qquad (8.5)$$

where

h_{max} = maximum wall thickness, mm

T_x = recommended part ejection temperature, °C

T_m = recommended mold temperature, °C

T_i = polymer injection temperature, °C

α = thermal diffusivity coefficient, mm^2/s

The data needed for making cooling time predictions are given in Table 8.5, which contains a list of the most widely injection-molded thermoplastics. It should be noted that Eq. (8.5) tends to underestimate the cooling time for very thin wall moldings. One reason is that for such parts the thickness of the runner system is often greater than the parts themselves and the greater delay is needed to ensure that the runners can be ejected cleanly from the mold. It is suggested that 3 s be taken as the minimum cooling time even if Eq. (8.5) predicts a smaller value.

The most important observation to make about Eq. (8.5) is that for a given polymer, with given molding temperatures, the cooling time varies with the square of the wall thickness of the molded part. This is the principal reason why injection molding is often uneconomical for thick-wall parts. It should also be noted that Eq. (8.5) applies only to a rectangular slab which is representative of the main wall of an injection-molded part. For a solid cylindrical section a correction factor of 2/3 should be used on the diameter since cooling takes place more rapidly for this boundary condition. Thus a 3 mm thick flat part with a 6 mm diameter cylindrical projection would have an equivalent maximum thickness of 2/3 × 6 = 4 mm.

8.7.3 Mold Resetting

Mold opening, part ejection and mold closing times depend upon the amount of movement required for part separation from the cavity and core and on the time required for part clearance from the mold plates during free fall. The summation of these three machine operation times is referred to as the resetting time. Approximate times for these machine operations have been suggested by Ostwald [12] for three general part shape categories: namely, flat, box shaped and deep cylindrical parts. These are given in Table 8.3.

It is clear that these estimates can only be viewed as very rough approximations since they do not include the effect of part size. The part size will influence resetting time in two ways. First, the projected area of the part together with the number of cavities will determine the machine size and hence the power available for mold opening and closing. Second, the depth of the part will, of course, determine the amount of mold opening required for part ejection.

In order to take account of the above factors, use will be made of injection molding machine data where typical maximum clamp strokes and dry cycle times for various sizes of molding machines are given. Dry cycle time is defined as the time required to operate the injection unit and then to open and close an appropriately sized mold by an amount equal to the maximum clamp stroke. Table 8.4 gives values of these parameters for a wide range of currently available injection molding machines.

It should be realized that the dry cycle time given by a machine supplier bears little relationship to the actual cycle time when molding parts. This is because the dry cycle time is based on an empty injection unit and it takes only milliseconds to inject air through the mold. Moreover, there is obviously no required delay for cooling and the machine clamp is operated during both opening and closing at maximum stroke and at maximum safe speed.

In practice the clamp stroke is adjusted to the amount required for the molding of any given part. If the depth of the part is given by D cm, then for the

Table 8.3 Machine Clamp Operation Times (s)

	Flat	Box	Cylindrical
Mold open	2	2.5	3
Part eject	0	1.5	3
Mold close	1	1	1

Table 8.4 Injection Molding Machines

Clamping force (kN)	Shot size (cc)	Operating cost ($h)	Dry cycle times (s)	Max. clamp stroke (cm)	Driving power (kW)
300	34	28	1.7	20	5.5
500	85	30	1.9	23	7.5
800	201	33	3.3	32	18.5
1100	286	36	3.9	37	22.0
1600	286	41	3.6	42	22.0
5000	2290	74	6.1	70	63.0
8500	3636	108	8.6	85	90.0

Table 8.5 Processing Data for Selected Polymers

Thermoplastic	Specific gravity	Thermal Thermal (mm²/s)	Injection temp. (°C)	Mold temp. (°C)	Ejection temp. (°C)	Inj'n pressure (bars)
High-density polyethylene	0.95	0.11	232	27	52	965
High-impact	1.59	0.09	218	27	77	965
Acrylonitrile-butadiene-styrene (ABS)	1.05	0.13	260	54	82	1000
Acetal (homopolymer)	1.42	0.09	216	93	129	1172
P{olyamide (6/6 nylon)	1.13	0.10	291	91	129	1103
Polycarbonate	1.20	0.13	302	91	127	1172
Polycarbonate (30% glass)	1.43	0.13	329	102	141	1310
Modified polyphenylene oxide (PPO)	1.06	0.12	232	82	102	1034
Modified PPO (30% glass)	1.27	0.14	232	91	121	1034
Polypropylene (40% talc)	1.22	0.08	218	38	88	965
Polyester teraphthalate (30% glass)	1.56	0.17	293	104	143	1172

present time estimation purposes it will be assumed that the clamp stroke is adjusted to a value of 2D + 5 cm. This will give the mold opening required for complete separation of the part from the cavity and matching core with a clearance of 5 cm for the part to fall away.

It can be noted from Table 8.3 that mold opening usually takes place more slowly than mold closing. This is because, during mold opening, ejection of the part takes place usually with a significant level of force to separate the part from the core onto which it will have shrunk. Rapid mold opening may thus result in warping or fracture of the molded part. For present time estimation purposes it will be assumed that opening takes place at 40 percent of the closing speed; this corresponds to the average of Ostwald's data in Table 8.3.

The precise motion of a clamp unit depends upon the clamp design and its adjustment. To obtain a simple estimate of resetting time it will be assumed that for a given clamp unit the velocity profile during a clamp movement (opening or closing) will have identical shape irrespective of the adjusted stroke length. Under these conditions, the time for a given movement will be proportional to the square-root of the stroke length.

Thus if the maximum clamp stroke is L_s for a given machine and the dry cycle time is t_d, then the time for clamp closing at full stroke will be assumed equal to $t_d/2$. However, if a part of depth D is to be molded, then the adjusted clamp stroke will be 2D + 5 cm and the time for mold closing will be

$$t_{close} = 0.5\,t_d\,[(2D + 5)/L_s]^{1/2} \tag{8.6}$$

If we now use the assumption of 40 percent opening speed and a dwell of 1 s for the molded part to fall between the plates, then this gives an estimate for mold resetting as

$$t_r = 1 + 1.75\,t_d\,[(2D + 5)/L_s]^{1/2} \tag{8.7}$$

Examples of Resetting Time

Assume that plain, 15 cm diameter cylindrical cups, with a depth of 20 cm, are to be manufactured from ABS in a six-cavity mold. From the example in Sect. 8.5 we know that the appropriate machine size is 8,500 kN and from Table 8.4 the corresponding values of dry cycle time, t_d, and maximum clamp stroke, l_s are 8.6 s and 85 cm, respectively. Substituting the values of $D = 20$, $L_s = 85$, and $t_d = 8.6$ into Eq. (8.6) gives an estimated resetting time of 12.0 s.

If the depth of the cylindrical cups is 10 cm, then the estimate of resetting time changes to 9.2 s, while if 10 cm diameter disks with a thickness of only 3 mm are to be molded then Eq. (8.6) predicts the resetting time to be 4.9 s. These estimates can be seen to be at some variance with the Ostwald data, which were only reasonable averages for typical small parts.

8.8 MOLD COST ESTIMATION

The skills needed for mold design and construction differ substantially from those required for all the other steps in the injection molding process. As a consequence, mold design usually takes place in isolation from the various other functions involved. This presents a serious hurdle to the exchange of information and ideas between the tool maker and molder on one side, and the part designer on the other. Desirable changes in part design often become evident only after major investments in tooling and testing have already been made. The consequences of such a belated recognition can be very significant in terms of final cost and part quality. On the other hand, mold cost estimations made during the concept design stage itself will help in identifying acceptable part and mold configurations before actual investment in the mold is made.

The mold cost can be broken down into two major categories: (a) the cost of the prefabricated mold base consisting of the required plates, pillars, guide bushings, etc., and (b) cavity and core fabrication costs. These will be discussed separately in the following sections.

8.8.1 Mold Base Costs

From a survey of currently available prefabricated mold bases, it has been shown by Dewhurst and Kuppurajan [13] that mold base cost is a function of the surface area of the selected mold base plates and the combined thickness of the cavity and core plates. Figure 8.7 shows mold base costs plotted against a single parameter based on area and thickness values. The data in Fig. 8.7 can be represented by

$$C_b = 1000 + 0.45\,A_c h_p^{0.4} \tag{8.8}$$

where

C_b = cost of mold base, $

A_c = area of mold base cavity plate, cm^2

h_p = combined thickness of cavity and core plates in mold base, cm

The selection of an appropriate mold base is based on the depth of the part, its projected area and the number of cavities required in the mold. In addition to the cavity size, extra allowance has to be given for molds with mechanical action side-pulls and other complicated mechanisms such as unscrewing devices for the molding of screw threads.

To determine the appropriate mold base size for a particular part, it is necessary to imagine the molded part (or parts for a multicavity operation) embedded within the mold base plates. The part(s) must have adequate clearance from the plate surfaces (and from each other) to provide the necessary rigidity against distortion from the cavity pressure during molding and to allow

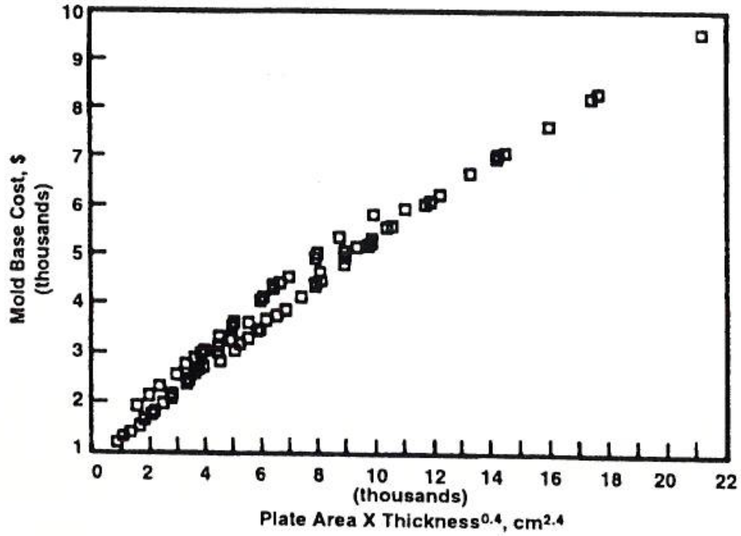

Figure 8.7 Principal mold base cost driver.

space for cooling channels and any moving core devices. Typically the minimum clearance between adjacent cavities and between cavity surfaces and the edges and rear surfaces of cavity plates should be 7.5 cm. The extra plate size required to accommodate side-pulls and unscrewing devices will depend upon the actual mechanisms used. However, for the purpose of the present cost-estimating procedure, it will be assumed that side-pulls or side unscrewing devices require twice the minimum clearance from the edges while rear unscrewing devices require a doubling of the material at the rear of the cavity. Thus one side-pull will increase the plate width or length by an additional 7.5 cm. Additional side-pulls will result in further plate size increases so that four or more pulls, one or more on each side of a part, will require a plate which is 15 cm larger in both length and width. It should also be noted that two side-pulls restricts the mold design to a single row of cavities while three or more usually implies single-cavity operation.

Example:

As an example of applying the above rules, assume that 10 cm diameter plain cylindrical cups with a depth, h_d, of 15 cm are to be molded in a six-cavity mold. A 3 × 2 array of cavities with clearances as specified above gives the required plate area A_c is 2550 cm^2. The combined cavity and core plate thickness h_p is $h_d + 15 = 30$ cm. Hence the mold base cost parameter $A_c h_p^{0.4}$ is 9940 cm$^{2.4}$ and applying this value to the graph in Fig. 8.7 leads to an estimated mold base cost of \$5,500.

If a more complex cylindrical part of the same size is imagined, with two diametrically opposed holes in the side surfaces and an internal thread, the estimated plate size increases will be as follows. The cavity plate will now hold a single row of six cavities in order to accommodate the diametrically opposed side pulls. Using 15 cm clearance along each side of the cavities to house the side core mechanisms, the plate area is 112.5 × 40 or 4500 cm^2. To support the unscrewing device, the combined plate thickness increases to an assumed value of 37.5 cm, which results in a new value of $A_c h_p^{0.4}$ equal to 19,179 cm$^{2.4}$. Referring to Fig. 8.7, this corresponds to a new mold base cost of approximately \$9,500.

It should be noted that mold makers will often increase the clearances between cavities as the cavity area increases. From assessment of a large number of molds, it seems the typical clearance may increase by about 0.5 cm for every 100 cm^2 of cavity area. This rule of thumb would have a marginal effect on the estimated mold base costs in the above example. However, it should be applied for larger parts to obtain a better estimate of mold base size and cost. The above costs are only for the mold base with square flat plates. The costs to manufacture the necessary cavities and moving cores is the subject of the next section.

8.8.2 Cavity and Core Manufacturing Costs

Initial cost estimates in the present work will be based on the use of a standard two-plate mold. The decisions regarding the use of three-plate molds, hot runner systems, etc., can only be made by comparing the increased cost of the mold system with the reduced machine supervision associated with semiautomatic or fully automatic operation.

As discussed in the last section, mold making starts with the purchase of a pre-assembled mold base from a specialist supplier. The mold base includes the main plates, pillars, bushings, etc. However, in addition to the manufacture of the cavities and cores, a substantial amount of work has to be performed on the mold base in order to transform it into a working mold. The main tasks are the deep hole drilling of the cooling channels and the milling of pockets in the plates to receive the cavity and core inserts. Additional tasks are associated with custom work on the ejector plate and housing to receive the ejection system, the insertion of extra support pillars where necessary and the fitting of electrical and coolant systems. A rule of thumb [14] in mold manufacture is that the purchase price of the mold base should be doubled to account for the custom work which has to be performed on it.

Determination of the cost of an injection mold involves a knowledge of the number of ejector pins to be used. This information would not usually be available at the early stages in part design. Discussions with experienced mold makers have indicated that the number of ejector pins is governed by such factors as the size of the part, the depth of the main core, the depth and closeness of ribs, and other features contributing to part complexity. However, analysis of a range of parts for which the corresponding number of ejector pins could be determined yielded no strong relationships between number of pins and part depth, part size or part complexity. The closest relationship was found to be based simply on the projected cross-sectional area of the parts at right angles to the direction of molding. With some considerable scatter, the number of ejector pins used was found to be approximately equal to the square root of the cross-sectional area when measured in square centimeters, i.e.,

$$N_e = A_p^{0.5} \qquad (8.9)$$

where

$N_e =$ number of ejector pins required

$A_p =$ projected part area, cm^2

Equation (8.9) will be used in the estimation of the cost of the ejection system for a molded part. An investigation of mold making costs by Sors et al. [15] suggests an approximate value of 2.5 manufacturing hours for each ejector pin, and this will be used in the present work. From Eq. (8.9) this gives the additional number of manufacturing hours for the ejection system of a part as

$$M_e = 2.5 \times A_p^{0.5}\ \mathrm{h} \qquad (8.10)$$

It is recognized that part ejection is not always accomplished through the use of ejector pins. Nevertheless, Eq. (8.10) represents a reasonable basis for estimating the cost of an ejection system at the concept design stage.

The geometric complexity of a part to be molded is handled in the present mold cost estimation scheme by assigning a complexity score on the range 0 to 10 for both the inner and outer surface of the part. The number of mold-manufacturing hours, associated with the geometrical features of the part, for one cavity and matching core(s) is then estimated from

$$M_x = 45\,(X_i + X_o)^{1.27}\ \mathrm{h} \qquad (8.11)$$

where X_i and X_o are the inner and outer complexity of the part, respectively.

This empirical relationship was obtained by Archer [16] from analysis of a wide range of injection-molded parts from small brackets to large cabinets and items of furniture.

It is expected that for rapid cost estimating a quick judgment can be made as to the appropriate complexity numbers. However, in order to gain confidence in the assignment of the different levels of geometrical complexity, a simple complexity counting procedure has been established as described below.

Geometrical Complexity Counting Procedure

Count all separate surface segments on the part inner surface. The inner surface is the surface which is in contact, during molding, with the main core and other projections or depressions in the core plate. Surface segments are either planar or have constant or smoothly changing curvature. The junction of different surface segments can be a sudden change (discontinuity) in either slope or curvature. Second, count the number of holes and depressions in the part wall on the inner surface. The complexity of the inner surface is given by

$$X_i = 0.01\,N_{sp} + 0.04\,N_{hd} \qquad (8.12)$$

where

$N_{sp} =$ number of surface patches

$N_{hd} =$ number of holes and depressions

The above procedure is repeated for the part outer surface to obtain the outer surface complexity level. Through holes should, of course, not be counted again from the outer surface. Also when counting surface patches, small connecting blend surfaces should not be counted.

When counting multiple identical features on the surface of a part, a power index of 0.7 should be used to account for the savings of machining identical features in the mold. For example, if the surface of a part is covered by 100 spherical dimples, then the equivalent number of surface patches to be counted is $100^{0.7} = 25$.

Example:

A plane conical component with recessed base is to be injection-molded; see Fig. 8.8. The inner and outer surface complexity levels are established as follows. The inner surface comprises the following surface segments:

1. Main conical surface
2. Flat base

Thus

$$X_i = 0.01 \times 2 = 0.02$$

The outer surface comprises:

1. Main conical surface
2. Flat annular base
3. Cylindrical recess in the base
4. Flat recessed base

In addition, the outer surface has the single depression in the base: so

$$X_o = 0.01 \times 4 + 0.04 \times 1 = 0.08$$

In addition to geometrical complexity, the size of the part to be molded clearly also affects the cost of the cavity and core inserts. Building on a part area relationship given by Sors et al. [15], Archer [16] has shown, from analysis of a wide range of injection molds, that for parts with very simple

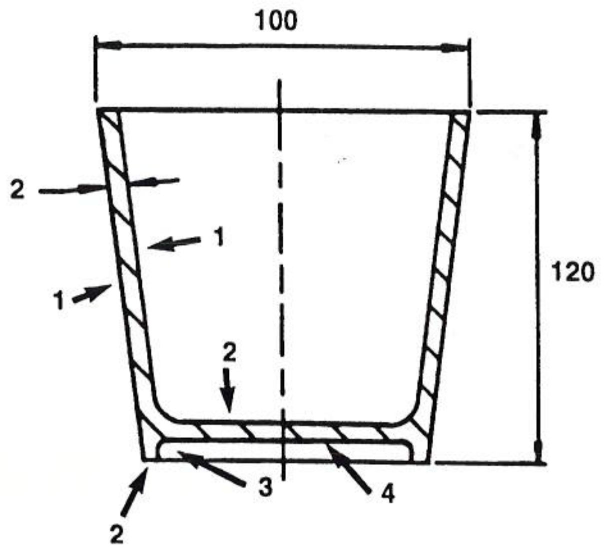

Figure 8.8 Surface segments of plain conical component (all dimensions in mm).

geometry the number of manufacturing hours for one cavity and core can be represented by

$$M_{po} = 5 + 0.085 \times A_p^{1.2}\,h \qquad (8.13)$$

where

A_p = part projected area, cm^2

The sum of the point scores from Eqs. (8.10), (8.11) and (8.13) provides a base estimate of the number of manufacturing hours to make one cavity and core and the ejection system for a part of given size with a known degree of geometrical complexity. However, in order to complete a mold cost estimating system six additional important factors need to be considered. These are:

(a) The need for retractable side-pulls or internal core lifters
(b) The requirement for one or more unscrewing cores to produce molded screw threads
(c) The surface finish and appearance specified for the part
(d) The average tolerance level applied to the part dimensions
(e) The requirement for one or more surfaces to be textured, e.g., checkered, leathergrain finish, etc.

(f) The shape of the surface across which the cavity and core separate: referred to in die design as the parting line

From discussion with a number of mold makers, it appears that the slideways and associated angle pins or withdrawal mechanisms for a side-pull, excluding manufacture of the core, will have an associated manufacturing time of 50 to 80 h. However, constructing an internal mechanism in the main core (sometimes called a lifter) to retract an internal core pin is substantially more difficult and may take between 100 and 200 h. More difficult still is the building of an unscrewing mechanism for the molding of screw threads, which may require 200 to 300 tool-making hours. For the present, early costing procedure manufacturing hours for side-pulls, internal lifters or unscrewing devices will be assumed to correspond to the average of these estimates.

The incorporation of texture into mold cavity surfaces is usually carried out by specialist companies that offer a wide range of standard texture patterns. It appears that the cost of texturing is proportional to both the complexity and size of the part and that a fairly good estimate is obtained by allowing 5 percent of the basic cavity manufacturing cost. Shallow lettering which can be etched or engraved into the mold can be considered equivalent to texture and costed in the same way.

The hand finishing of cavities required to produce high-quality surfaces on molded parts is extremely costly and time consuming. The time involved is clearly dependent on the size of the cavity, its geometrical complexity and the required appearance of the molded part. In this context it is necessary to differentiate between opaque and transparent parts. For opaque parts, the required appearance can be separated into four categories: not critical, standard (Society of Plastics Engineers No. 3 finish), high gloss (SPE No. 2) and highest gloss (SPE No. 1). On the other hand, transparent parts are generally produced according to only two categories; standard finish and with some internal flaws permissible, or highest quality with internal blemishes unacceptable. These two categories are more difficult to achieve than categories two and four respectively, for opaque parts. From discussions with mold makers, it appears that the time taken to finish a cavity and core to achieve the above appearance levels can be represented as a percentage increase applied to the basic time for cavity and core manufacture. For the present estimating system, this translates into a percentage increase to the sum of the manufacturing hours predicted by Eqs. (8.11) and (8.13). Reasonable percentage values for the different part appearance categories are given in Table 8.6.

The tolerances which are given to the dimensions of an injection-molded part must clearly be within the capabilities of the process. These capabilities will be addressed later in this chapter. However, the part tolerances also indirectly affect the cost of mold manufacture. The reason for this is that the mold maker will be required to work within a small portion of the part toler-

Table 8.6 Percentage Increases for Different Appearance Levels

Appearance	Percentage increase
Not critical	10
Opaque, standard (SPE #3)	15
Transparent, standard internal flaws or waviness permissible	20
Opaque, high gloss	25
Transparent, high quality	30
Transparent, optical quality	40

ances in order to leave the remainder of the tolerance bands to cover variations in the molding process. Tighter tolerances will thus result in more careful cavity and core manufacture. Evidence from mold makers suggests that this effect, while less significant than surface finish requirements, depends on the number of features and dimensions rather than on the part size. In terms of the present cost estimating procedure, the part tolerance affects the time estimate for geometrical complexity given by Eq. (8.11). Acceptable percentage increases, which should be applied to the result of Eq. (8.11), for the six different tolerance levels are given in Table 8.7.

The final consideration is the shape of the plane separating the cavity and core inserts. Whenever possible, the cavity and core inserts should be mounted in flat opposing mold plates. This results in a flat parting plane (straight parting line) which only requires surface grinding to produce a well-fitting mold. Flat bent parts, or hollow parts whose edge, separating the inner and outer surface, does not lie on a plane, cannot be molded with a flat parting plane. For these cases, the parting surface should be chosen from the six classifications given in Table 8.8. For each of these separate categories, industrial data suggest that the additional number of manufacturing hours required to manufacture the mold is approximately proportional to the square root of the cavity area as given by the following relationship.

Table 8.7 Percentage Increase for Tolerance

Tolerance level	Description of tolerances	Percentage increase
0	All greater than ±0.5 mm	0
1	Most approx. ±0.35 mm	2
2	Most approx. ±0.25 mm	5
3	Several approx. ±0.25 mm	10
4	Several approx. ±0.05 mm	20
5	Most approx. ±0.05 mm	30

$$M_s = f_p A_p^{1/2} h \tag{8.14}$$

where

A_p = projected area of cavity, cm^2

f_p = parting plane factor given in Table 8.8

M_s = additional mold manufacturing hours for nonflat parting surface

8.9 MOLD COST POINT SYSTEM

Following the above discussions, a point system for mold cavity and core cost estimating can now be established. The main cost drivers will simply be listed in order and associated graphs or tables will be referred to for determination of the appropriate number of points. The mold manufacturing cost is determined by equating each point to one hour of mold manufacture.

(i) Projected Area of Part (cm^2)
—refer to Eqs. (8.10) and (8.13), which include points for the size effect on manufacturing cost plus points for an appropriate ejection system

(ii) Geometric Complexity

Table 8.8 Parting Surface Classification

Parting surface type	Factor (f_p)
Flat parting plane	0
Canted parting surface or one containing a single step	1.25
2-4 simple steps or a simple curved surface	2
Greater than 4 simple steps	2.5
Complex curved surface	3
Complex curved surface with steps	4

—identify complexity ratings for inner and outer surfaces on scale of 0 to 10 according to the procedure described earlier

—apply Eq. (8.11) to determine the appropriate point score

(iii) Side-Pulls

—identify number of holes or apertures requiring separate side-pulls (side cores) in the molding operation

—allow 65 points for each side-pull

(iv) Internal Lifters

—identify number of internal depressions or undercuts requiring separate internal core lifters

—allow 150 points for each lifter

(v) Unscrewing Devices

—identify number of screw threads which would require an unscrewing device

—allow 250 points for each unscrewing device

(vi) Surface Finish/Appearance

—refer to Table 8.6 to identify the appropriate percentage value for the required appearance category

—apply the percentage value to the sum of the points determined for (i) and (ii) to obtain the appropriate point score related to part finish and appearance

(vii) Tolerance Level

—refer to Table 8.7 to identify the appropriate percentage value for the required tolerance category

—apply the percentage value to the geometrical complexity points determined for (ii) above to obtain the appropriate point score related to part tolerance

(viii) Texture

—if portions of the molded part surface require standard texture patterns, such as checkered, leather grain, etc., then add 5 percent of point scores from (i) and (ii)

(ix) Parting Plane

—determine the category of parting plane from Table 8.8 and note the value of the parting plane factor, f_p

—use f_p to obtain the point score from Eq. (8.14)

To determine the cost to manufacture a single cavity and matching core(s) the total point score is multiplied by the appropriate average hourly rate for tool manufacture.

Example:

It is anticipated that 2,000,000 plain hollow conical components are to be molded in Acetal homopolymer. The component, illustrated in Fig. 8.8, has a material volume of 78 cm^3 and a projected area in the direction of molding of 78.5 cm^2. The mold manufacturing points are first established as follows:

		Points
(i)	Projected Area of Part (substitute $A_p = 78.5\ cm^2$ into Eqs. (8.10 43 and 8.13)	43
(ii)	Geometrical Complexity (established earlier as $C_i = 0.02$ and $C_o = 0.08$ for this part—apply Eq. (8.11) for points)	2
(iii)	Number of Side-Pulls	0
(iv)	Number of Internal Lifters	0
(v)	Number of Unscrewing Devices	0
(vi)	Surface Finish/Appearance (Opaque high gloss; see Table 8.6—add 25% of 43 + 2)	11
(vii)	Tolerance Level (category l; see Table 8.7—insignificant effect for low complexity)	0
(viii)	Texture	0
(ix)	Parting Plane (category 0)	0
	Total point score =	56

Assuming an average rate of $40 per hour for mold manufacturing, the estimated cost for one cavity and core is found to be 56 x $40, or $2,240.

8.10 ESTIMATION OF THE OPTIMUM NUMBER OF CAVITIES

A major economic advantage of injection molding is its ability to make multiple parts in one machine cycle through the use of multicavity molds. Sometimes the cavities in a mold may be for different parts which are to be used together in the same product. This type of mold is referred to as a family mold. It is used infrequently since it requires the existence of a family of parts, made of the same material, and having similar thicknesses. It also has the obvious disadvantage of individual reject parts always requiring the remanufacture of the entire family.

The common practice is the use of multicavity molds to make sets of identical parts with each molding cycle. The motivation in this case is to reduce processing cost through an initially higher mold investment. The effect of the chosen number of cavities on part cost can be dramatic. This means that the cost of alternative designs of a particular part can only be estimated if the appropriate number of cavities is known. The identification of the appropriate number of cavities for a particular part will be explored in this section.

When a multicavity mold is used, three principal changes occur:

(i) A larger machine with a greater hourly rate is needed than would be the case for a single-cavity mold.
(ii) The cost of the mold is clearly greater than for a single-cavity one.
(iii) The manufacturing time per part decreases in approximately inverse proportion to the number of cavities.

In order to identify the optimum number of cavities, the increase in hourly rate with machine size increase must be known. Also an estimate must be available of the cost of a multicavity mold compared to the cost of a single-cavity mold for the same part. With regard to the first requirement, Fig. 8.9 shows a national survey of injection molding machine rates which was carried out by *Plastics Technology Magazine* [17]. It can be seen that the hourly rate can be represented almost precisely as a linear relationship based on the machine clamp force; i.e., machine hourly rate is

$$C_r = k_1 + m_1 F \text{ \$/h} \tag{8.15}$$

where

F = clamp force, kN
k_1, m_1 = machine rate coefficients

For the latest machine rate data, obtained by the authors, and given in Table 8.4, $k_1 = 25\$/h$ and $m_1 = 0.0091\$/h/kN$.

Turning now to the cost of multicavity mold manufacture, evidence in the literature [18] suggests that the cost of multiple cavity and core inserts, compared with the cost of one unique cavity/core set, follows an approximate power law relationship. That is, if the cost of one cavity and matching core is given by C_1, then the cost, C_n, of producing identical sets of the same cavity and core can be represented by

$$C_n = C_1 n^m \tag{8.16}$$

where m is a multicavity mold index and n is the number of identical cavities.

Testing of this relationship for a wide range of multicavity molds suggests that a reasonable value of m for most molding applications is 0.7. This value can be interpreted as an approximate rule that doubling the number of cavity and core inserts will always involve a cost increase of 62 percent since $2^{0.7}$ is 1.62.

Savings also occur in the mold base cost per cavity when increasing the number of cavities. This can readily be established from the discussion of mold base costs in the previous section. However, the savings depend upon the cavity area, smaller cavities being associated with larger savings. Nevertheless, to allow a simple analysis to be performed, it will be assumed that a power law relationship similar to Eq. (8.16) applies equally to mold bases and with the same value for the power index. With this assumption the cost of a complete production mold satisfies the same relationship; i.e.,

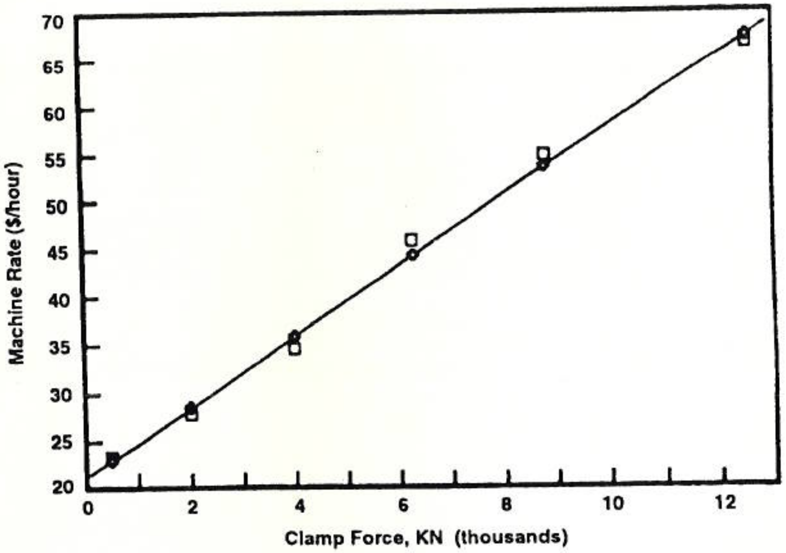

Figure 8.9 National average injection molding machine rates.

$$C_{cn} = C_{c1} n^m \tag{8.17}$$

where

C_{c1} = cost of single-cavity mold
C_{cn} = cost of n-cavity mold
n = number of cavities
m = multicavity mold index

Using the above relationship, the cost, C_t, of producing N_t molded components can be expressed as

$$\begin{aligned} C_t &= \text{processing cost} + \text{mold cost} + \text{polymer cost} \\ &= (N_t/n)(k_1 + m_1 F)t + C_{c1} n^m + N_t C_{pm} \end{aligned} \tag{8.18}$$

where

t = machine cycle time, h

C_m = cost of polymer material per part, \$

Assuming that an infinite variety of different clamp force machines were available, then one with just sufficient force would be chosen since hourly rate increases with clamp force (Fig. 8.3). Thus we can write

$$F = nf \tag{8.19}$$

where

f = separating force on one cavity

Substituting Eq. (8.19) into (8.18) gives

$$C_t = N_t(k_1f/F + m_1f)t + C_{c1}(F/f)^m + N_tC_m \tag{8.20}$$

For a given part, molded in a particular polymer, of which N_t are required, the only variable in Eq. (8.20) is clamp force F. Thus the minimum value of C_t will occur when dC_t/dF is equal to zero or when

$$-N_tk_1ft/F^2 + mC_{c1}F^{(m-1)}/f^m = 0 \tag{8.21}$$

Multiplying Eq. (8.21) by F and substituting $n = F/f$ gives the expression for the optimum number of cavities as

$$n = (N_tk_1t/(mC_{c1}))^{1/(m+1)} \tag{8.22}$$

A number of simplifying assumptions were used to derive Eq. (8.22) and for this reason the value predicted for the optimum number of cavities should be regarded as a first approximation. However, Eq. (8.22) is very easily applied and provides a reasonable basis for comparing alternative designs during the concept phase of a new injection-molded part.

8.11 DESIGN EXAMPLE

Figure 8.10 shows an injection molded cover which is currently manufactured by a U.S. automobile company. The cover has a flange which has thickened pads at locations around the periphery where bolts secure it to the assembly. The main body of the part is 2 mm thick and the bolt pads have a thickness of 4.6 mm.

The design can be considered to be produced from a 2 mm thick basic shape, referred to in injection molding jargon as the main wall. Features are then added to the main wall. In the present design the features are:

(i) The eight triangular stiffening ribs (called gussets) which support each pad to the side wall
(ii) The six through holes

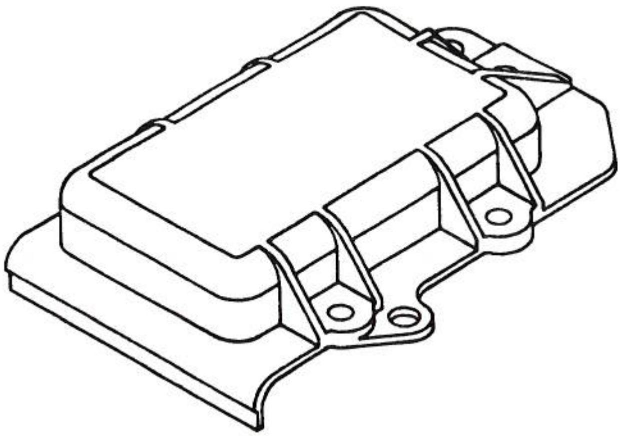

Figure 8.10 Heater core cover.

(iii) The four pads of thickness 2.6 mm which rest on top of the main wall to give a combined thickness of 4.6 mm

Adding features to the main wall will always increase the mold cost. In addition, if they result in an increase in wall thickness, then the cycle time will also increase.

Using the point system in Sec. 8.8, it can readily be shown that for the cover design the increase in mold cost due to all eighteen features is approximately 23 percent, or \$1,150, and that the cost of one cavity and core set would be approximately \$8,000. Thus for a fairly modest production volume of 50,000 parts the added mold cost is only 2.3 cent or 0.13 cents per feature.

In contrast, the cooling equation in Sec. 8.6 shows that the addition of the thickened pads will increase the cycle time by 110 percent, which corresponds to an increase in processing cost per part of 12.3 cents.

Thus if we wish to reduce the cost of the cover, the obvious way is to reduce the material thickness in the stiffened areas around the bolt clearance holes. The way in which this can be achieved is through the use of ribbed structures as shown in Fig. 8.11. The projecting circular rib around each hole is known as a boss and this is supported by a network of intersecting straight ribs as shown. The rib structure can be deep enough to give equivalent stiffness to the solid 4.6 mm thick pads. Also, the recommended rib thickness would be 2/3 of the main wall thickness, or 1.67 mm. For a production volume of 50,000 parts the new design would have an associated cavity and core cost of approximately \$10,500 corresponding to an increase in mold cost per part of 5 cents. However, this would be more than offset by the 12.3 cents decrease in

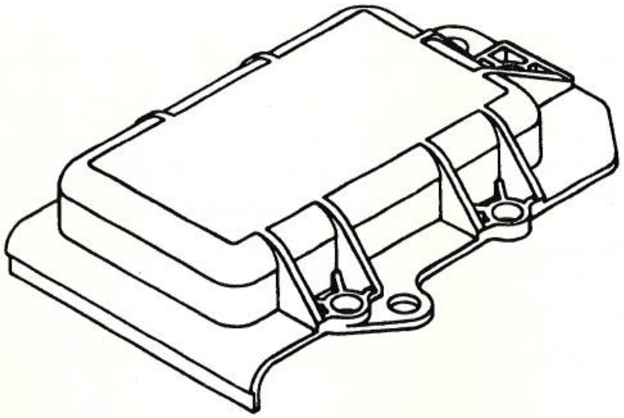

Figure 8.11 Proposed redesign of heater core cover.

processing cost. Just as important, the new design is likely to result in high-quality, distortion-free moldings. The existing design, in contrast, is difficult to mold because of the different wall thicknesses. The problems are due to the continued cooling of the pads after the main wall has fully solidified. This results in a buildup of residual or locked-in stresses as the pad material continues to shrink while constrained by the surrounding solidified wall.

8.12 INSERT MOLDING

Insert molding refers to the common practice of molding small metal items, such as pins or bushings, into injection molded parts. Most typically, the inserts are hand-loaded into the mold cavity, or cavities, prior to closing of the mold and activation of the injection unit. For this reason, insert molding machines are of a vertical design so that inserts can be placed into a horizontal cavity plate when the mold is open. Machine rates can be assumed to be the same as for conventional molding machines. An approximate estimate of the cost of an insert molded part can be obtained by adding 2 s per insert to the molding cycle time. Thus for a four-cavity mold, with two inserts per cavity, the cycle time would be increased by 16 s.

For high-volume manufacture of inserted molded parts, special-purpose machines can be obtained which employ multiple stations. The simplest type has a shuttle table which moves two separate molds alternately into the molding machine. With this system, inserts can be loaded into one mold while the other one is being filled and allowed to cool. Following this procedure, insert loading takes place within the machine cycle but with a higher cost machine and a larger mold investment.

It should be noted that insert molding does not find universal favor. Many manufacturing engineers feel that the high risk of mold damage due to misplaced inserts offsets any advantage of the process. The alternative is simply to mold the depressions needed to accept inserts, which can then be secured later by ultrasonic welding. This process is described briefly later in the chapter.

8.13 DESIGN GUIDELINES

Suppliers of engineering thermoplastics have in general provided excellent support to the design community. Several have published design manuals or handbooks which are required reading for those designing injection molding components. Information can be obtained from them on the design of ribbed structures, gears, bearings, spring elements, etc.

The interested reader may wish to write, in particular, to Du Pont, G.E. Plastics Division or Mobay Corporation for design information associated with their engineering thermoplastics.

Generally accepted design guidelines are listed below.

1. Design the main wall of uniform thickness with adequate tapers or draft for easy release from the mold. This will minimize part distortion by facilitating even cooling throughout the part.
2. Choose the material and the main wall thickness for minimum cost. Note that a more expensive material with greater strength or stiffness may often be the best choice. The thinner wall which this choice allows will reduce material volume to offset the material cost increase. More important, the thinner wall will significantly reduce cycle time and hence processing cost.
3. Design the thickness of all projections from the main wall with a preferred value of 1/2 of the main wall thickness and do not exceed 2/3 of the main wall thickness. This will minimize cooling problems at the junction between the projection and main wall where the section is necessarily thicker.
4. Preferably align projections in the direction of molding or at right angles to the molding direction lying on the parting plane. This will eliminate the need for mold mechanisms.
5. Avoid depressions on the inner side surfaces of the part which would require moving core pins to be built inside the main core. The mechanisms to produce these movements (referred to in mold making as lifters) are very expensive to build and maintain. Through holes on the side surfaces, instead of internal depressions, can always be produced with less expensive side-pulls.
6. If possible, design external screw threads so that they lie in the molding

plane. Alternatively, use a rounded or rolled-type thread profile which can be stripped from the cavity or core without rotating. In the latter case, polymer suppliers should be consulted for material choice and appropriate thread profile and depth.

In addition to the above general rules, design books should be consulted for design tips and innovative design ideas. Many of these are concerned with methods for producing undercuts and side features without the need for mold mechanisms. This will be explored a little further in the next section when snap fit elements are discussed.

One important cautionary note should be made with regard to design guidelines. Guidelines should never be followed when doing so may have a negative effect on the cost or quality of the assembly as a whole. This applies particularly to the guidelines aimed at avoiding mechanisms in the mold. The only valid rule in this regard is that the need for mold mechanisms should be recognized by the designer, and if they are unavoidable, then their cost should be justified in the early stages of design. This cost may be simply the cost of the mechanisms or it may also include the increase in processing cost if they restrict the number of cavities below the optimum value. The worst case is a high-volume component with side-pulls or unscrewing devices on all sides so that it must be made uneconomically in a single-cavity mold.

8.14 ASSEMBLY TECHNIQUES

One of the major advantages of injection molding is its ability to easily incorporate, in the molded parts, effective self-securing techniques. In the present context, self-securing refers to the ability to achieve a secure assembly without the use of separate fasteners or the addition of a separate bonding agent. Two of these self-securing techniques are also widely used with metal parts; namely, press fitting and riveting. With press fitting much larger interferences are possible with injection-molded parts than is the case with metal parts. This has the advantage of requiring less precise tolerance control for the press fit. The negative aspect of plastic press fit joints is that the material is constantly under stress and will invariably relax over a period of time to produce some degradation of the joint strength. Testing of press fits under the expected loading conditions is therefore essential. With regard to riveting, integral rivets are of course easily produced through inexpensive feature additions to the mold. On assembly the rivet heads are also easily formed by cold heading or by the use of heated forming tools; the latter operation is sometimes referred to as staking.

A third self-securing method, which is unique to plastic parts, is the use of ultrasonic welding. This is a method of joining two or more plastic molded parts through the generation of intermolecular frictional heat at the assembly interfaces. Ultrasonic welding equipment simply involves a special fixture to hold and clamp the parts and through which a high-frequency vibration of approximately 20 kHz is passed. The detail design of the butting or overlapping joints is critical for successful joining of the parts. Figure 8.12 shows typical recommended joint designs. Ultrasonic welding is a good economic choice where sealed joining is required, since the equipment is relatively inexpensive and the process is fast. Welding is accomplished typically in about 2 s.

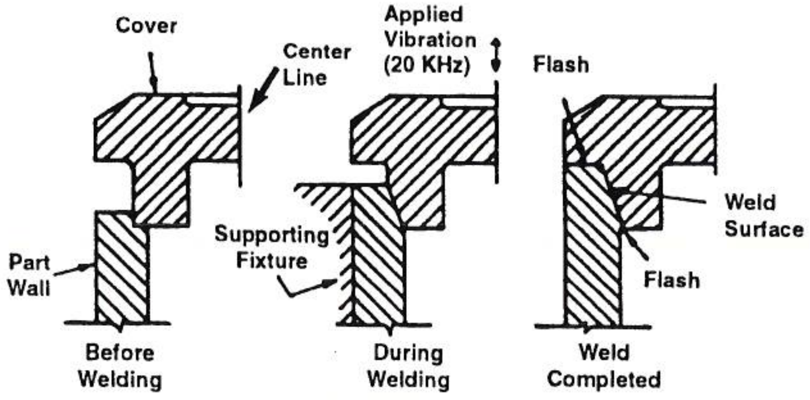

Figure 8.12 Ultrasonic welding joint design by Du Pont.

The final and most widely recognized self-securing method for molded parts is through the use of snap fit elements. These can be separated into two main

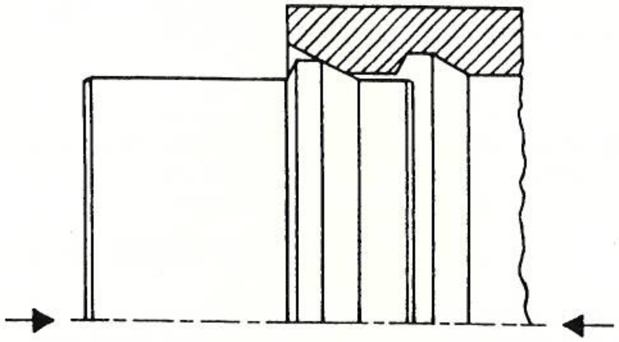

Figure 8.13 Annular snap joint design.

types. The first type was developed for mating parts of circular cross-section and involves the use of a cylindrical undercut and mating lip as shown in Fig. 8.13. The male partner of the mating pair may be molded with the parting plane along its axis with very little added cost, simply the cost of adding the groove feature to the cavity and core geometrics. However, if the male cylinder cannot be designed at right angles to the direction of mold opening, then its separation from the mold will require the use of side-pulls. In contrast, the female or undercut part is almost always stripped off the core by the mold ejection system and is, therefore, inexpensively produced. Snap fits of this type are only truly satisfactory for circular parts. The further part shapes deviate from circular, the more difficult it becomes to eject parts from the mold which can be assembled satisfactorily.

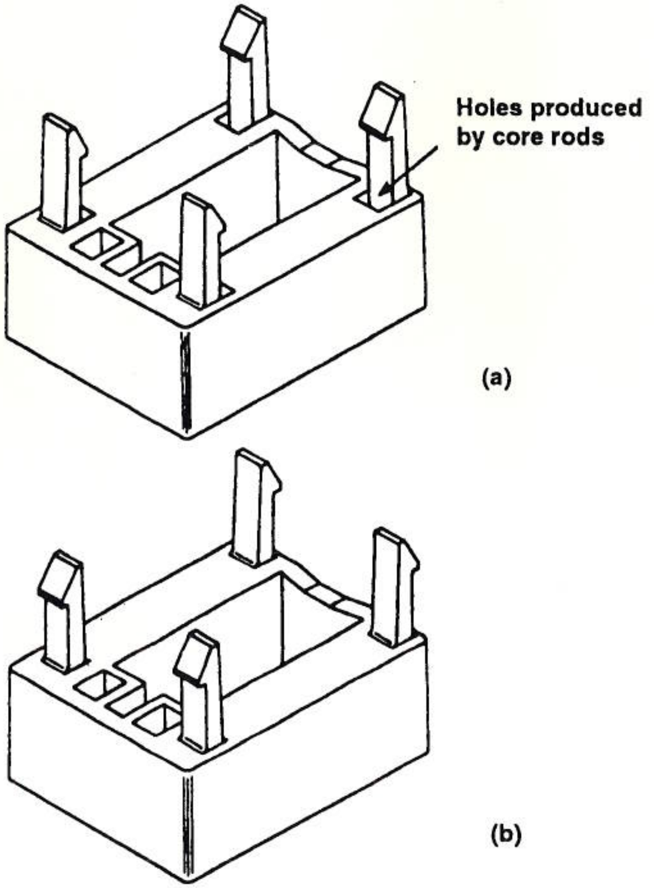

Figure 8.14 Cantilever snap fit elements: (a) undercuts formed by core rods. (b) undercuts formed by side-pulls.

The second type of snap fit design involves the use of one or more cantilever snap elements as shown in Fig. 8.14. If possible, the cantilever snap element and its mating undercut should be designed for molding without mold mechanisms; one such design is shown in Fig. 8.14a. The alternative design, illustrated in Fig. 8.14b will require a side-pull for the molding of the cantilever and an expensive core lifter to mold the undercut. However, even in the latter case, for a large enough production volume the extra mold costs can be easily outweighed by the subsequent savings in assembly cost.

Polymer manufacturers should be contacted for detailed information on snap fit design.

REFERENCES

1. Dewhurst, P. and Boothroyd, G. Design for Assembly in Action, Assembly Engineering, Jan. 1987.
2. Rosato, D.V., (Editor), Injection Molding Handbook, Van Nostrand Reinhold Co., NY, 1986.
3. Bown, J., Injection Molding of Plastic Components, McGraw-Hill Book Company (UK) Limited, 1979.
4. MacDermott, C.P., Selecting Thermoplastics for Engineering Applications, Marcel Dekker Inc., NY, 1984.
5. Bernhardt, E.C., (Editor), Computer-Aided Engineering for Injection Molding, Hanser Publishers, Munich, 1983.
6. Design Handbook for Dupont Engineering Polymers, E.I. du Pont de Nemours and Co. Inc., 1986.
7. Farrell, R.E., "Injection Molding Thermoplastics," Modern Plastics Encyclopedia, pp. 252-270, 1985-86.
8. Khullar, P., "A Computer-Aided Mold Design Systems for Injection Molding of Plastics," Ph.D. Dissertation, Cornell University, 1981.
9. Gordon Jr., B.E., "Design and Development of a Computer Aided Processing System with Application to Injection Molding of Plastics," Ph.D. Thesis, W.P.I. Worcester, MA, November 1976.
10. Carslaw, H.S and Jaeger, J.C., Conduction of Heat in Solids, Oxford, Clarendon Press, 1986.
11. Ballman, P. and Shusman, R., Easy Way to Calculate Injection Molding Set-Up Time, Modern Plastics, McGraw-Hill, NY, 1959.
12. Ostwald, P.F., (Editor), American Cost Estimator, American Machinist, McGraw-Hill, NY, 1985.
13. Dewhurst, P. and Kuppurajan, K., Optimum Processing Conditions for Injection

Molding, Report No. 12, Product Design for Manufacture Series, University of Rhode Island, February 1987.
14. Schuster, A., Injection Mold Tooling, Society of Plastic Engineers Seminar, New York, Sept. 30–Oct. 1, 1987.
15. Sors, L., Bardocz, L. and Radnoti,I., Plastic Molds and Dies, Van Nostrand Reinhold Co., 1981.
16. Archer, D., Economic Model of Injection Molding, M.S. Thesis, URI, 1988.
17. Plastics Technology Magazine, June 1987.
18. Reinbacker, W.R., A Computer Approach to Mold Quotations, PACTEC V., 5th Pacific Technical Conference, Los Angeles, February 1980.