University of Jordan
Department of Chemical Engineering
Polymers and Plastics Engineering (0905553)

Processing of Plastics

2) Injection Moulding

One of the most common processing methods for plastics is injection moulding. Nowadays every home, every vehicle, every office, every factory contains a multitude of different types of articles which have been injection moulded. These include such things as electric drill casing, yoghurt cartons, television housings, combs, syringes, paint brush handles, crash helmets, gearwheels, typewriters, fascia panels, reflectors, telephones, brief cases – the list is ead1ess.

The original injection moulding machines were based on the pressure die casting technique for metals. The first machine is reported to have been patented in the United States in 1872, specifically for use with Celluloid. This was an important invention but probably before its time because in the following years very few developments in injection moulding processes were reported and it was not until the 1920s, in Germany, that a renewed interest was taken in the process. The first German machines were very simple pieces of equipment and relied totally on manual operation. Levers were used to clamp the mould and inject the melted plastic with the result that the pressures which could be attained were not very high. Subsequent improvements led to the use of pneumatic cylinders for clamping the injection which not only lifted some of the burden off the operator but also meant that higher pressures could be used.

The next major development in injection moulding, i.e. the introduction of hydraulically operated machines, did not occur until the late 1930s when a wide range of thermoplastics started to become available. However, these machines still tended to be hybrids based on die casting technology and the design of injection moulding machines for plastics was not taken really seriously until the 1950s when a new generation of equipment was developed. These machines catered more closely for the particular properties of polymer melts and modern machines are of the same basic design although of course the control systems are very much more sophisticated nowadays.

In principle, injection moulding is a simple process. A thermoplastic, in the form of granules or powder, passes from a feed hopper into the barrel where it is heated so that it becomes soft. It is then forced through a nozzle into a relatively cold mould which is clamped tightly closed. When the plastic has had sufficient time to become solid the mould opens, the article is ejected and the cycle is repeated. The major advantages of the process include its versatility in moulding a wide range of products, the ease with which automation can be introduced, the possibility of high production rates and the manufacture of articles with close tolerances. The basic injection moulding concept can also be adapted for use with thermosetting materials.

Details of the Process

The earliest injection moulding machines were of the plunger type as illustrated in Fig. 1 and there are still many of these machines in use today. A pre-determined quantity of moulding material drops from the feed hopper into the barrel. The plunger then conveys the material along the barrel where it is heated by conduction from the external heaters. The material is thus plasticized under pressure so that it may be forced through the nozzle into the mould cavity. In order to split up the mass of material in the barrel and improve the heat transfer, a torpedo is fitted in the barrel as shown.

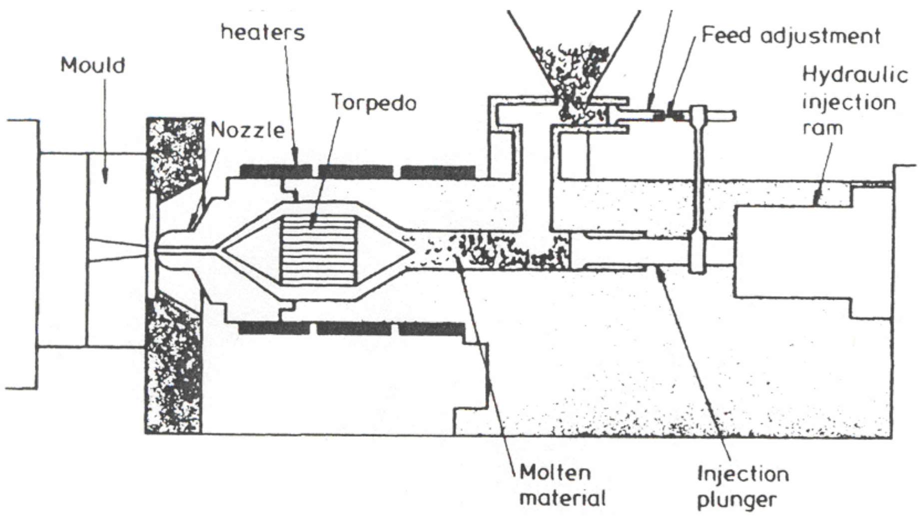


Fig 1. Plunger type injection machine

Unfortunately there are a number of inherent disadvantages with this type of machine which can make it difficult to produce consistent moulding. The main problems are:

(a) There is little mixing or homogenization of the molten plastic.

(b) It is difficult to meter accurately the shot size. Since metering is on a volume basis, any variation in the density of the material will alter the shot weight.

(c) Since the plunger is compressing material which is in a variety of forms (Varying from a solid granule to a viscous melt) the pressure at the nozzle can vary quite considerably from cycle to cycle.

(d) The presence of the torpedo causes a significant pressure loss.

(e) The flow properties of the melt are pressure sensitive and since the pressure is erratic, this amplifies the variability in mould filling.

Some of the disadvantages of the plunger machine may be overcome by using a pre-plasticizing system. This type of machine has two barrels. Raw material is fed into the first barrel where an extruder screw or plunger plasticizes the material and feeds it through a non-return valve into the other barrel. A plunger in the second barrel then forces the melt through a nozzle and into the mould. In this system there is much better homogenization because the melt has to pass though the small opening connecting the two barrels. The shot size can also be metered more accurately since the volume of material fed to the second barrel can be controlled by a limit switch on its plunger. Another advantage is that there is no longer a need for the torpedo on the main injection cylinder.

However, nowadays this type of machine is seldom used because it is considerably more complicated and more expensive than necessary. One area of application where it is still in use is for large mouldings because a large volume of plastic can be plasticized prior to injection using the primary cylinder plunger.

For normal injection moulding, however, the market is now dominated by the reciprocating screw type of injection moulding machine. This was a major breakthrough in machine design and yet the principle is simple. An extruder type screw in a heated barrel performs a dual role. On the one hand it rotates in the normal way to transport, melt and pressurize the material in the barrel but it is also capable, whilst not rotating, of moving forward like a plunger to inject melt into the mould. A typical injection moulding machine cycle is illustrated in Fig. 2. It involves the following stages:

(a) After the mould closes, the screw (not rotating) pushes forward to inject melt into the cooled mould Fig. 2.a. The air inside the mould will be pushed out through small vents at the furthest extremities of the melt flow path.

(b) When the cavity is filled, the screw continues to push forward to apply a holding pressure (see Fig. 2.b). This has the effect of squeezing extra melt into the cavity to compensate for the shrinkage of the plastic as it cools. This holding pressure is only effective as long as the gate(s) remain open.

(c) Once the gate(s) freeze, no more melt can enter the mould and so the screw-back commences , Fig. 2.c. At this stage the screw starts to rotate and draw in new plastic from the hopper. This is conveyed to the front of the screw but as the mould cavity is filled with plastic, the effect is to push the screw backwards. This prepares the next shot by accumulating the desired amount of plastic in front of the screw. At a pre-set point in time, the screw stops rotating and the machine sits waiting for the solidification of the moulding and runner system to be completed.

(d) When the moulding has cooled to a temperature where it is solid enough to retain its shape, the mould opens and the moulding is ejected. The mould then closes and the cycle is repeated (see Fig. 3).

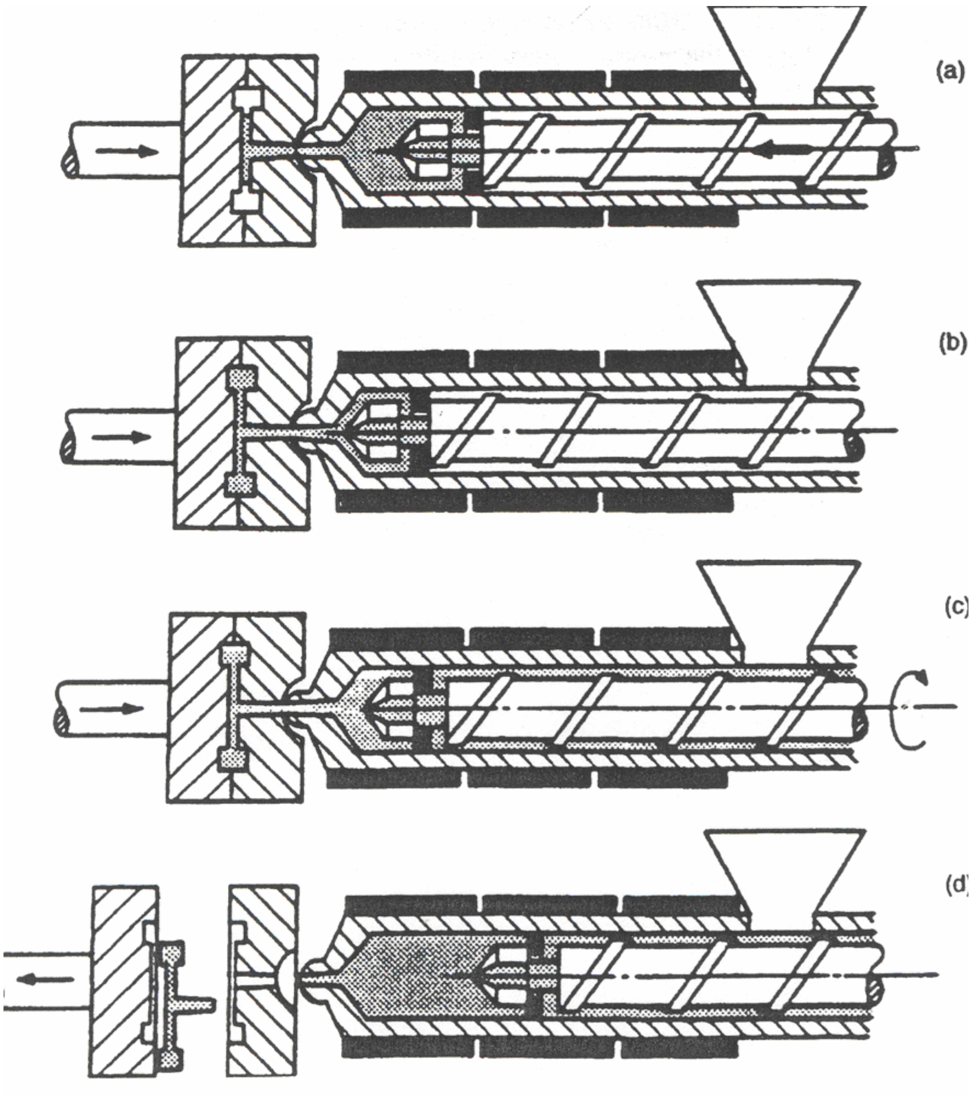


Fig. 2 Typical cycle in reciprocating screw injection moulding machine

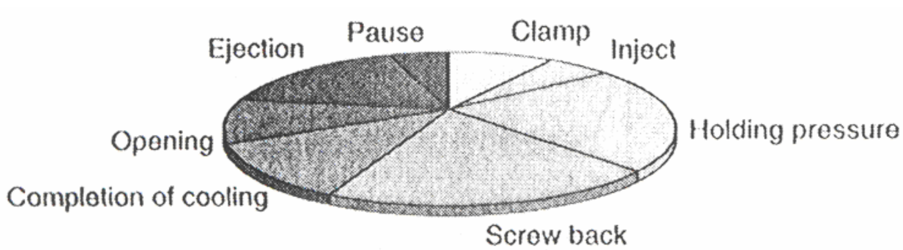
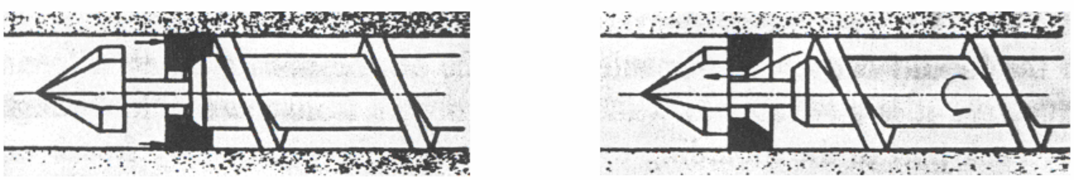


Fig.3 Stages during injection moulding

There are a number of important features in reciprocating screw injection moulding machines and these will now be considered in turn:

Screws

The screws used in these machines are basically the same as those described for extrusion. The compression ratios are usually in the range 2.5:1 to 4:1 and the most common L/D ratios are in the range 15 to 20. Some screws are capable of injecting the plastic at pressures up to 200 MN/m^2. One important difference from an extruder screw is the presence of a back-flow check valve at the end of the screw as illustrated in Fig. 4. The purpose of this valve is to stop any back flow across the flights of the screw when it is acting as a plunger. When material is being conveyed forward by the rotation of the screw, the valve opens as shown. One exception is when injection moulding heat-sensitive materials such m PVC. In such cases there is no check valve because this would provide sites where material could get clogged and would degrade.



(a) Valve closed (b) Valve open

Fig. 4 Typical check valve

Barrel and Heaters

These are also similar to those in extruder machines. In recent years, vented barrels have become available to facilitate the moulding of water sensitive plastics without the need for pre-drying (PET, Nylon, and Polycarbonate). Water sensitivity in plastics can take several forms. If the plastic absorbs water then dimensional changes will occur. The plastic will also be plasticized by the water so that there will be property changes such as a reduction in modulus and an increase in toughness. All these effects produced by water absorption are reversible.

Another event which may occur is hydrolysis. This is a chemical reaction between the plastic and water. It occurs extremely slowly at room temperature but can be significant at moulding temperatures. Hydrolysis causes degradation, reduction in properties (such as impact strength) and it is irreversible.

Nozzles

The nozzle is screwed into the end of the barrel and provides the means by which the melt can leave the barrel and enter the mould. It is also a region where the melt can be heated both by friction and conduction from a heater band before entering the relatively cold channels in the mould. Contact with the mould causes heat transfer from the nozzle and in cases where this is excessive it is advisable to withdraw the nozzle from the mould during the screw-back part of the moulding cycle. Otherwise the plastic may freeze off in the nozzle.

There are several types of nozzle. The simplest is an open nozzle as shown in Fig. 5.a. This is used whenever possible because pressure drops can be minimized and there are no hold-up points where the melt can stagnate and decompose. However, if the melt viscosity is low then leakage will occur from this type of nozzle particularly if the barrel/nozzle assembly retracts from the mould each cycle. The solution is to use a shut-off nozzle of which there are many types. Fig. 5.b shows a nozzle which is shut off by external means. Fig. 5.c shows a nozzle with a spring loaded needle valve which opens when the melt pressure exceeds a certain value or alternatively when the nozzle is pressed up against the mould. Most of the shut-off nozzles have the disadvantage that they restrict the flow of the material and provide undesirable stagnation sites. For this reason they should not be used with heat sensitive materials such as PVC.

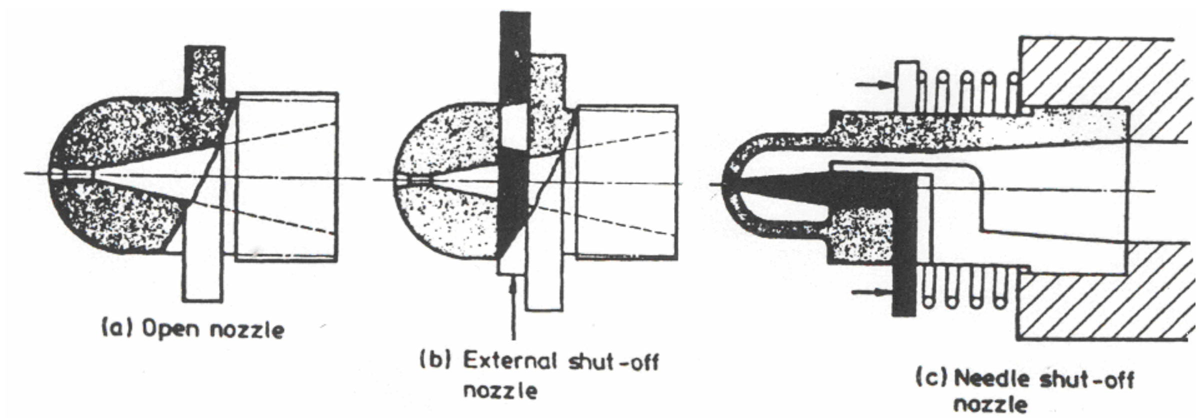


Fig. 5 Types of nozzle

Clamping Systems

In order to keep the mould halves tightly closed when the melt is being injected under high pressures it is necessary to have a clamping system. This may be either (a) hydraulic or (b) mechanical (toggle) – or some combination of the two.

In the hydraulic system, oil under pressure is introduced behind a piston connected to the moving platen of the machine. This causes the mould to dose and the clamp force can be adjusted so that there is no leakage of molten plastic from the mould.

The toggle is a mechanical device used to amplify face. Toggle mechanisms tend to be preferred for high speed machines and where the clamping face is relatively small.

Moulds

In the simplest case an injection mould (or ‘tool’) consists of two halves into which the impression of the part to be moulded is cut. The mating surfaces of the mould halves are accurately machined so that no leakage of plastic can occur at the split line. If leakage does occur the flash on the moulding is unsightly and expensive to remove. A typical injection mould is illustrated in Fig. 6. It may by seen that in order to facilitate mounting the mould in the machine and cooling and ejection of the moulding, several additions are made to the basic mould halves. Firstly, backing plates permit the mould to be bolted on to the machine platens. Secondly, channels are machined into the mould to allow the mould temperature to be controlled. Thirdly, ejector pins are included to that the moulded part can be freed from the mould. In most cases the ejector pins are operated by the shoulder screw hitting a stop when the mould opens. The mould cavity is joined to the machine nozzle by means of the sprue. The sprue anchor pin then has the function of pulling the sprue away from the nozzle and ensuring that the moulded part remains on the moving half of the mould, when the mould opens. For multi-cavity moulds the impressions are joined to the sprue by runners – channels cut in one or both halves of the mould through which the plastic will flow without restriction. A narrow constriction between the runner and the cavity allows the moulding to be easily separated from the runner and sprue. This constriction is called the gate

A production injection mould is a piece of high precision engineering manufactured to very close tolerances by skilled craftsmen. A typical mould can be considered to consist of (i) the cavity and core and (ii) the remainder of the mould (often referred to as the bolster).

Finishing and polishing the mould surfaces is extremely important because the melt will tend to reproduce every detail on the surface of the mould. The mould will have to be hardened to make it stand up to the treatment it receives in service. As a result of all the time and effort which goes into mould manufacture, it is sometimes found that a very complex mould costs more than the moulding machine on which it is used.

(a) Gates: As mentioned earlier the gate is the small orifice which connects the runner to the cavity. It has a number of functions. Firstly, it provides a convenient weak link by which the moulding can be broken off from the runner system. In some moulds the degating may be automatic when the mould opens. The gate also acts like a valve in that it allows molten plastic to fill the mould but being small it usually freezes off first. The cavity is thus sealed off from the runner system which prevents material being sucked out of the cavity during screw-back. As a general rule, small gates are preferable because no finishing is required if the moulding is separated cleanly from the

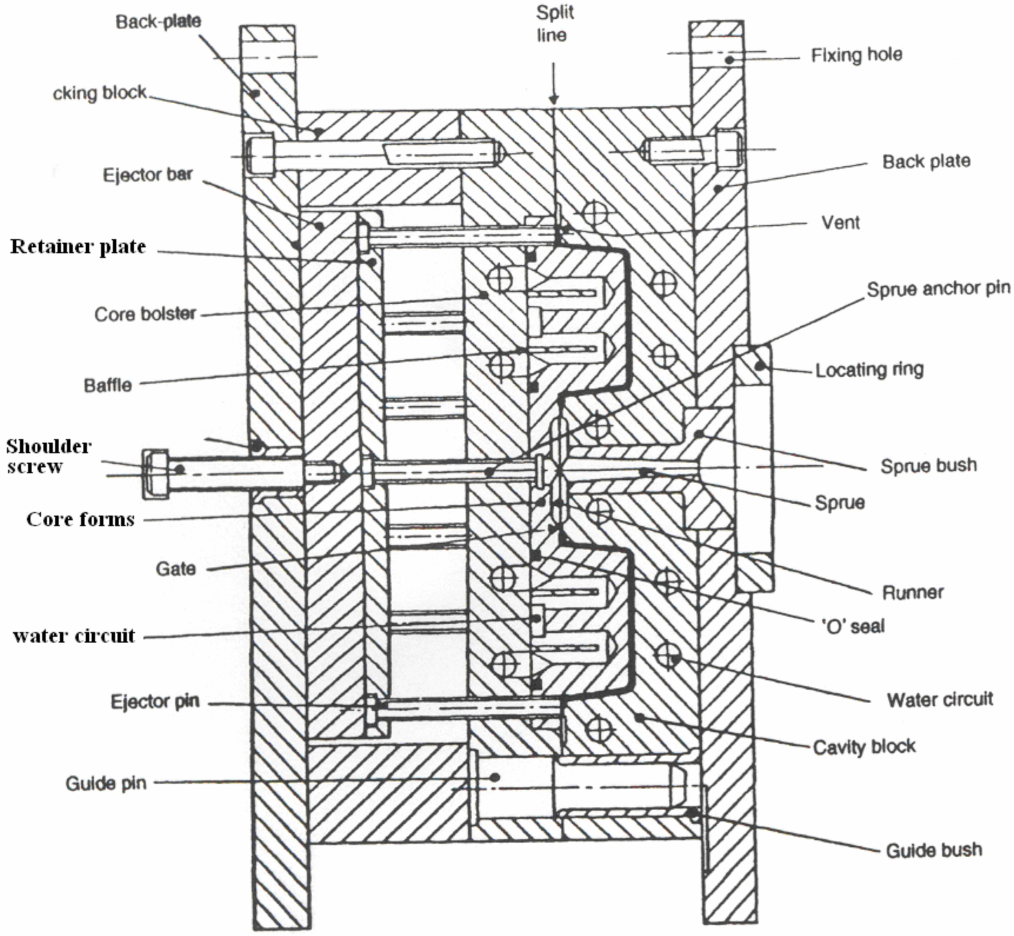


Fig. 6 Details of injection mould

runner. So for the initial trials on a mould the gates are as small as possible and are only opened up if there are mould filling problems.

In a multi-cavity mould it is not always possible to arrange for the runner length to each cavity to be the same. This means that cavities close to the sprue would be filled quickly whereas cavities remote from the sprue receive the melt later and at a reduced pressure. To alleviate this problem it is common to use small gates close to the sprue and progressively increase the dimensions of the gate further along the runners. This has the effect of balancing the fill of the cavities. If a single cavity mould is multi-gated then here again it may be beneficial to balance the flow by using various gate sizes.

Examples of gates which are in common use are shown in Fig. 7. Sprue gates are used when the sprue bush can feed directly into the mould cavity as, for example, with single symmetrical moulding such as buckets. Pin gates are particularly successful because they cause high shear rates which reduce the viscosity of the plastic and so the mould fills more easily. The side gate is the most common type of gate and is a simple rectangular section feeding into the side of the cavity. A particular attraction of this type of gate is that mould filling can be improved by increasing the width of the gate but the freeze time is unaffected because the depth is unchanged.

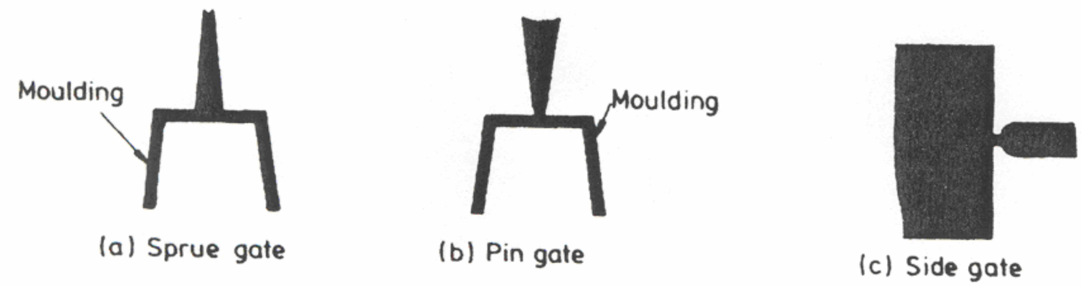


Fig. 7 Types of gate

(b) Runners: The runner is the flow path by which the molten plastic travels from the sprue (i.e. the moulding machine) to the gates (i.e. the cavity). To prevent the runner freezing off prematurely, its surface area should be small so as to minimize heat transfer to the mould. However, the cross sectional area of the runner should be large so that it presents little resistance to the flow of the plastic but not so large that the cycle time needs to be extended to allow the runner to solidify for ejection. A good indication of the efficiency of a runner is, therefore, the ratio of its cross-sectional area to its surface area. For example, a semi-circular channel cut into one half of the mould is convenient to machine but it only has an area ratio of 0.153 D where D is the diameter of the semi-circle. A full round runner, on the other hand, has a ratio of 0.25 D. A square section also has this ratio but is seldom used because it is difficult to eject. A compromise is a trapezoidal section (cut into one half of the mould) or a hexagonal section.

(c) Sprues: The sprue is the channel along which the molten plastic first enters the mould. It delivers the melt from the nozzle to the runner system. The sprue is incorporated in a hardened steel bush which has a seat designed to provide a good seal with the nozzle. Since it is important that the sprue is pulled out when the mould opens it is tapered as shown in Fig. 6 and there is a sprue pulling device mounted directly opposite the sprue entry. This can take many forms but typically it would be an undercut or reversed taper to provide a key for the plastic on the moving half of the mould. Since the sprue, like the runner system, is effectively waste it should not be made excessively long.

(d) Venting: Before the plastic melt is injected, the cavity in the closed mould contains air. When the melt enters the mould, if the air cannot escape it become compressed. At worst this may affect the mould filling, but in any case the sudden compression of the sir causes considerable heating. This may be sufficient to burn the plastic and the mould surface at local hot spots. To alleviate this

problem, vents are machined into the mating surfaces of the mould to allow the air to escape. The vent channel must be small so that molten plastic will not flow along it and cause unsightly flash on the moulded article. Typically a vent is about 0.025 mm deep and several millimeters wide. Away from the cavity the depth of the vent can be increased so that there is minimum resistance to the flow of the gases out of the mould.

(e) Mould Temperature Control: For efficient moulding, the temperature of the mould should be controlled and this is normally done by passing a fluid through a suitably arranged channel in the mould. The rate at which the moulding cools affects the total cycle time as well as the surface finish, tolerances, distortion and internal stresses of the moulded article. High mould temperatures improve surface gloss and tend to eliminate voids. However, the possibility of flashing is increased and sink marks are likely to occur. If the mould temperature is too low then the material may freeze in the cavity before it is filled. In most cases the mould temperatures used are a compromise based on experience.

Multi-Daylight Moulds

This type of mould, also often referred to as a three plate mould, is used when it is desired to have the runner system in a different plane from the parting line of the moulding. This would be the case in a multi-cavity mould where it was desirable to have a central feed to each cavity (see Fig. 8). In this type of mould there is automatic degating and the runner system and sprue are ejected separately from the moulding.

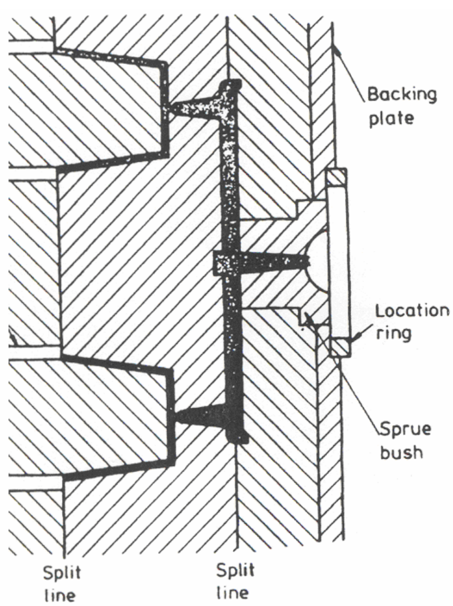


Fig. 8 Typical 3-plate mould

Hot Runner Moulds

The runners and sprues are necessary in a mould but they are not part of the end-product. Unfortunately, it is not economically viable to discard them so they must be re-ground for subsequent reprocessing. Regrinding is expensive and can introduce contamination into the material so that any system which avoids the accumulation of runners and sprues is attractive. A system has been developed to do this and it is really a logical extension of duce plate moulding. In this system, strategically placed heaters and insulation in the mould keep the plastic in the runner at the injection temperature. During each cycle therefore the component is ejected but the melt in the runner channel is retained and injected into the cavity during the next shot. A typical mould layout is shown in Fig. 9.

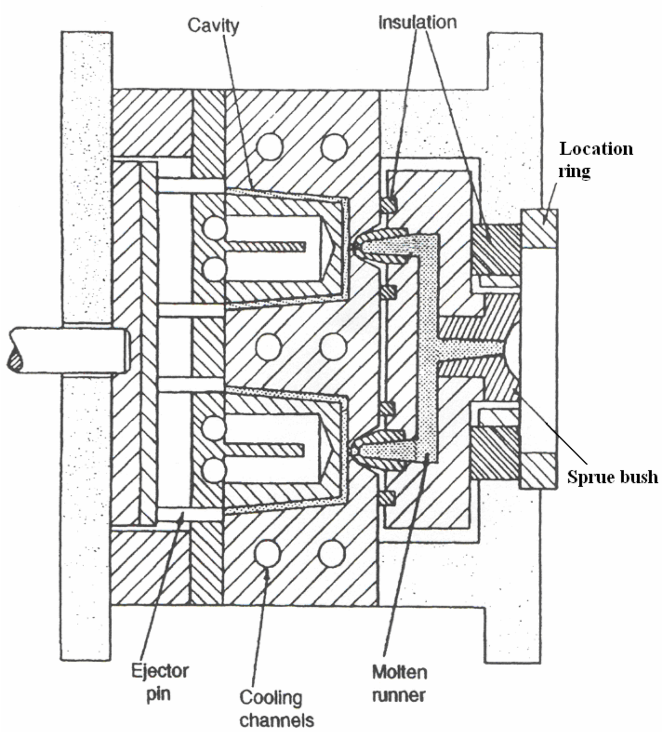


Fig. 9 Layout of hot runner mould

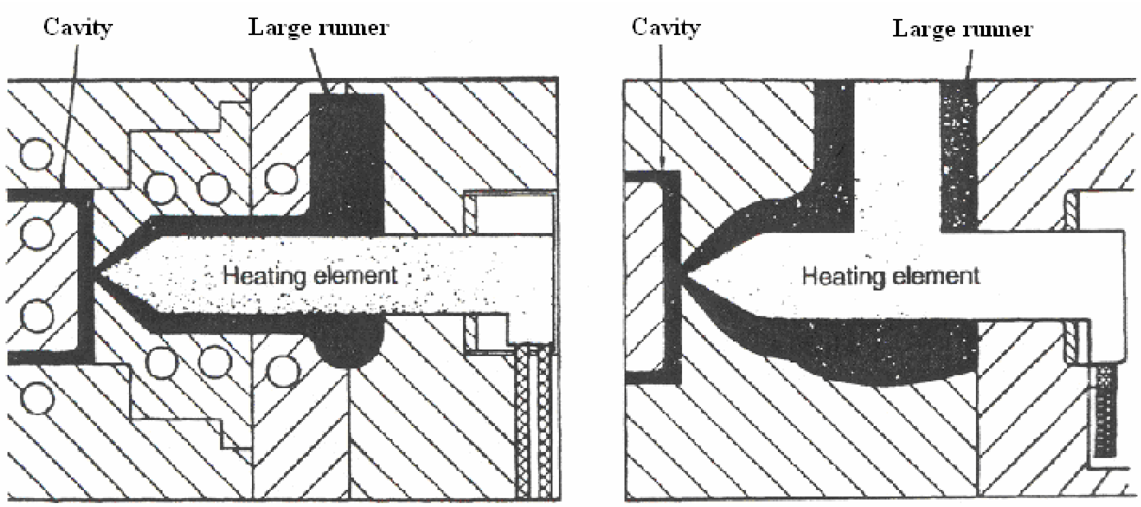
Additional advantages of hot runner moulds are (i) elimination of trimming and (ii) possibility of faster cycle times because the runner system does not have to freeze off. However, these have to be weighed against the disadvantages of the system. Since the hot runner mould is more complex than a conventional mould it will be more expensive. Also there are many areas in the hot runner manifold where material can get trapped. This means that problems can be experienced during colour or grade changes because it is difficult to remove all of the previous material. As a practical point it should also be realized that the system only works as long as the runner remains molten. If the runner system freezes off then the hot runner manifold needs to be dismantled to remove the runners. Note also that hot runner systems are not suitable for heat sensitive materials such as PVC.

Insulated Runner Moulds

This is similar in concept to the hot runner mould system. In this case, instead of having a specially heated manifold in the mould, large runners (13 – 25 mm diameter) are used. The relatively cold mould causes a frozen skin to form in the runner which then insulates its core so that this remains molten. As in the previous case the runner remains in the mould when the moulding is ejected and the molten part of the runner is then injected into the cavity for the next shot. If an undue delay causes the whole runner to freeze off then it may be ejected and when moulding is restarted the insulation layer soon forms again. This type of system is widely used for moulding of fast cycling products such as flower pots and disposable goods. The main disadvantage of the system is that it is not suitable for polymers or pigments which have a low thermal stability or high viscosity, as some of the material may remain in a semi-molten form in the runner system for long periods of time.

A recent development of the insulated runner principle is the distribution tube system. This overcomes the possibility of freezing-off by insertion of heated tubes into the runners. However, this system still relies on a thick layer of polymer forming an insulation layer on the wall of the runner and so this system is not suitable far heat sensitive materials.

Note that both the insulated runner and the distribution tube systems rely on a cartridge heater in the gate area to prevent premature freezing off at the gate (see Fig. 10).



(a) Insulated runner system (b) Distributed runner system

Fig. 10 Insulated and distributed runner systems

Mould Clamping Force

In order to prevent 'flashing', i.e. a thin film of plastic escaping out of the mould cavity at the parting line, it is necessary to keep the mould tightly closed during injection of the molten plastic. Before setting up a mould on a machine it is always worthwhile to check that there is sufficient clamping force available on the machine. To do this it is necessary to be able to estimate what clamping force will be needed. The relationship between mould area and clamp requirements has occupied the minds of moulders for many years. Practical experience suggests that the clamping pressure over the projected area of the moulding should be between 10 and 50 MN/m^2 depending on factors such as shape, thickness, and type of material.

Structural Foam Injection Moulding

Foamed thermoplastic articles have a cellular core with a relatively dense (solid) skin. The foam effect is achieved by the dispersion of inert gas throughout the molten resin directly before moulding. Introduction of the gas is usually carried out either by pre-blending the resin with a chemical blowing agent which releases gas when heated or by direct injection of the gas (usually nitrogen).

When the compressed gas/resin mixture is rapidly injected into the mould cavity, the gas expands explosively and forces the material into all parts of the mould.

The advantages of these types of foam moulding are:

(a) For a given weight they are many times more rigid than a solid moulding

(b) They are almost completely free from orientation effects and the shrinkage is uniform

(c) Very thick sections can be moulded without sink marks.

Foamed plastic articles may be produced with good results using normal screw-type injection moulding machines (see Fig. 11). However, the limitations on shot size, injection speed and platen area imposed by conventional injection equipment prevent the full large-part capabilities of structural foam from being realized. Specialized foam moulding machines currently in use can produce parts weighing in excess of 50 kg (see Fig. 12).

Wall sections in foam moulding are thicker than in solid material. Longer cycle times can therefore be expected due to both the wall thickness and the low thermal conductivity of the cellular material. In contrast, however, the injection pressures in foam moulding are low when compared with conventional injection moulding. This means that less clamping force is needed per unit area of moulding and mould costs are less because lower strength mould materials may be used.

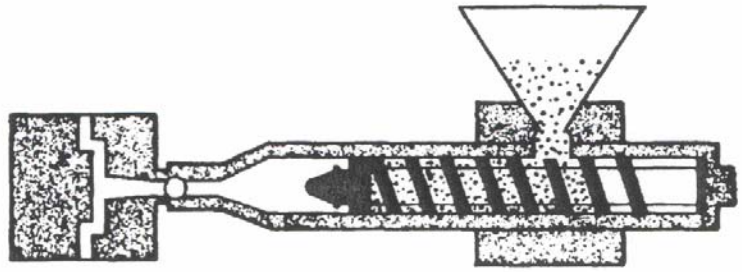


Fig. 11 Standard injection moulding press

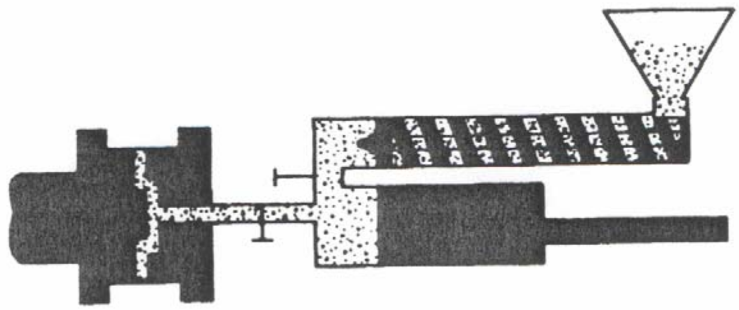


Fig. 12 Specialised foam moulding press

Sandwich Moulding

This is an injection moulding method which permits material costs to be reduced in large mouldings. In most mouldings it is the outer surface of an article which is important in terms of performance in service. If an article has to be thick in order that it will have adequate flexural stiffness then the material within the core of the article is wasted because its only function is to keep the outer surfaces apart. The philosophy of sandwich moulding is that two different materials (or two forms of the same material) should be used for the core and skin. That is an expensive high performance material is used for the skin and a low-cost commodity or recycled plastic is used for the core. The way that this can be achieved is illustrated in Fig. 13.

Initially the skin material in injected but not sufficient to fill the mould. The core material is then injected and it flows laminarly into the interior of the core. This continues until the cavity is filled as shown in Fig. 13.c. Finally the nozzle valve rotates so that the skin material is injected into the sprue thereby clearing the valve of core material in preparation for the next shot. In a number of cases the core material is foamed to produce a sandwich section with a thin solid skin and a cellular core.

It is interesting that in the latest applications of sandwich moulding it is the core material which is being regarded as the critical component. This is to meet design requirements for computers, electronic equipment and some automotive parts. In these applications there is a growing demand for covers and housings with electromagnetic interference (EMI) shielding. The necessity of using a plastic with a high loading of conductive filler (usually carbon black) means that surface finish is poor and unattractive. To overcome this the sandwich moulding technique can be used in that a good quality surface can be moulded using a different plastic.

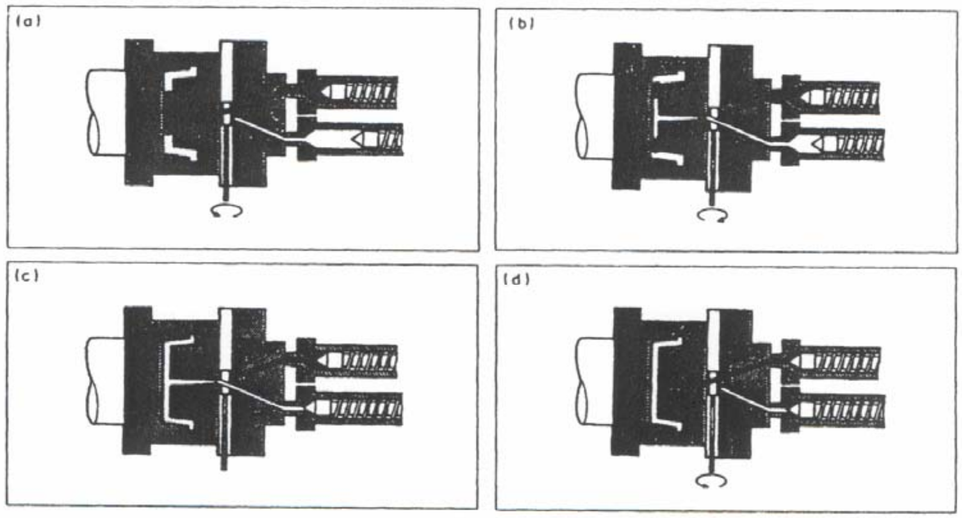


Fig. 13 Stages in sandwich moulding process

Gas Injection Moulding

In recent years major developments have been made in the use of an inert gas to act as the core in an injection moulded plastic product. This offers many advantages including greater stiffness/weight ratios and reduced moulded-in stresses and distortion.

The first stage of the cycle is the flow of molten polymer into the mould cavity through a standard feed system. Before this flow of polymer is complete, the injection of a predetermined quantity of gas into the melt begins through a special nozzle located within the cavity or feed system as shown in Fig. 14. The timing, pressure and speed of the gas injection are critical.

The pressure at the polymer gate remains high and, therefore, the gas chooses a natural path through the hotter and less viscous parts of the polymer melt towards the lower pressure areas. The flow of gas cores out a hollow centre extending from its point of entry towards the last point of fill. By controlling the amount of gas injected into the hollow core, the pressure on the cooling polymer is controlled and maintained until the moulding is packed. The final stage is the withdrawal of the gas nozzle, prior to mould opening, which allows the gas held in the hollow core to vent.

Whilst cycle times are comparable with those of conventional injection moulding, clamping forces are much lower. Also, by using gas to core out the polymer instead of mixing with it, gas-injection overcomes a number of shortcomings of the structural foam process. In particular there are no surface imperfections (caused by escaping gas bubbles in structural foam moulding) and are and cycle times are lower because thinner sections are being cooled.

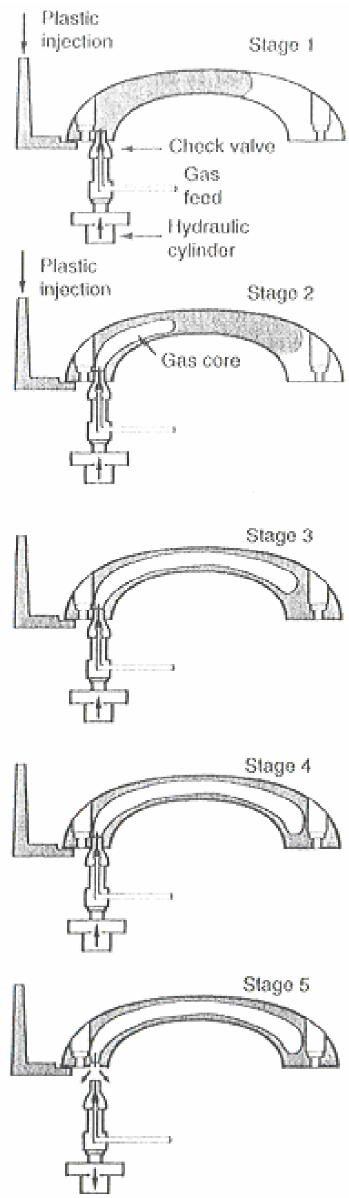


Fig. 14 Stages in the gas injection moulding of an automotive handle

Shear Controlled Orientation in Injection Moulding (SCORIM)

One of the major innovations in recent years is the use of pulsed pressure through the gates to introduce and control the orientation of the structure (or fillers) in injection moulded products. A special manifold is attached to the machine nozzle as illustrated in Fig. 15. This diagram relates to the double live feed of melt although up to four positions, capable of applying oscillating pressure may be used.

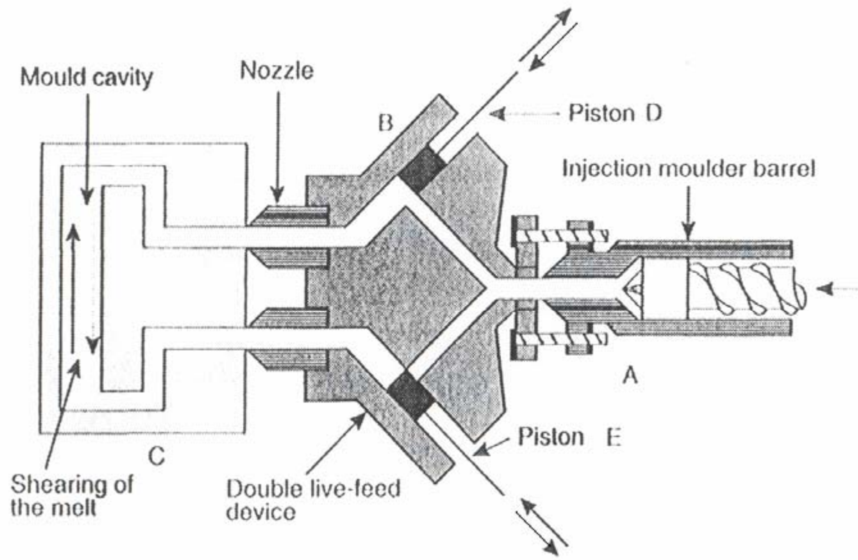


Fig. 15 One embodiment of SCORIM where the device (B) for producing shear during solidification, by the action of pistons (D) and (E), is placed between the injection moulding machine barrel (A) and the mould (C)

Shear controlled orientation in injection moulding (SCORIM) is based on the progressive application of macroscopic shears at the melt-solid interface during solidification in the moulding of a polymer matrix.

Macroscopic shears of specified magnitude and direction, applied at the melt-solid interface provide several advantages:

(i) Enhanced polymer matrix or fiber alignment by design in moulded polymers or fiber reinforced polymers.

(ii) Elimination of mechanical discontinuities that result from the initial mould filling process, including internal weld lines.

(iii) Reduction in the detrimental effects of a change in moulded section thickness.

(iv) Elimination or reduction in defects.

Reaction Injection Moulding

Although there have been for many years a number of moulding methods (such as hand lay-up of glass fibers in polyester and compression moulding of thermosets or rubber) in which the plastic material is manufactured at the same time as it is being shaped into the final article, it is only recently that this concept has been applied in an injection moulding type process. In Reaction

Injection Moulding (RIM), liquid reactants are brought together just prior to being injected into the mould. In-mould polymerization then takes place which forms the plastic at the same time as the moulding is being produced.

The basic RIM process is illustrated in Fig. 16. A range of plastics lend themselves to the type of fast polymerization reaction which is required in this process - polyesters. epoxies, nylons and vinyl monomers. However, by far the most commonly used material is polyurethane. The components A and B are an isocyanate and a polyol and these are kept circulating in their separate systems until an injection shot is required. At this point the two reactants are brought together in the mixing head and injected into the mould.

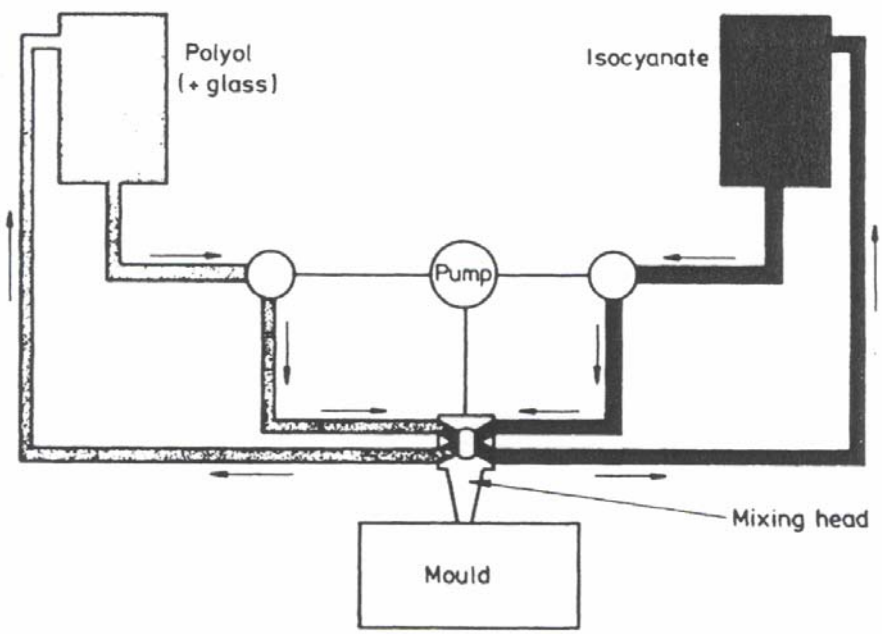


Fig. 16 Schematic view of reaction injection moulding

Since the reactants have a low viscosity, the injection pressures are relatively low in the RIM process. Thus, comparing a conventional injection moulding machine with a RIM machine having the same clamp force, the RIM machine could produce a moulding with a much greater projected area (typically about 10 times greater). Therefore the RIM process is particularly suitable for large area mouldings such as car bumpers and body panels. Another consequence of the low injection pressures is that mould materials other than steel may be considered. Aluminum has been used successfully and this permits weight savings in large moulds. Moulds are also less expensive than injection moulds but they must not be regarded as cheap. RIM moulds require careful design and, in particular, a good surface finish because the expansion of the material in the mould during polymerization causes every detail on the surface of the mould to be reproduced on the moulding.

Injection Blow Moulding

Previously we considered the process of extrusion blow moulding which is used to produce hollow articles such as bottles. At that time it was mentioned that if molecular orientation can be introduced to the moulding then the properties are significantly improved. In recent years the process of injection blow moulding has been developed to achieve this objective. It is now very widely used for the manufacture of bottles for soft drinks.

The steps in the process are illustrated in Fig. 17. Initially a preform is injection moulded. This is subsequently inflated in a blow mould in order to produce the bottle shape. In most cases the second stage inflation step occurs immediately after the injection moulding step but in some cases the preforms are removed from the injection moulding machine and subsequently re-heated for inflation.

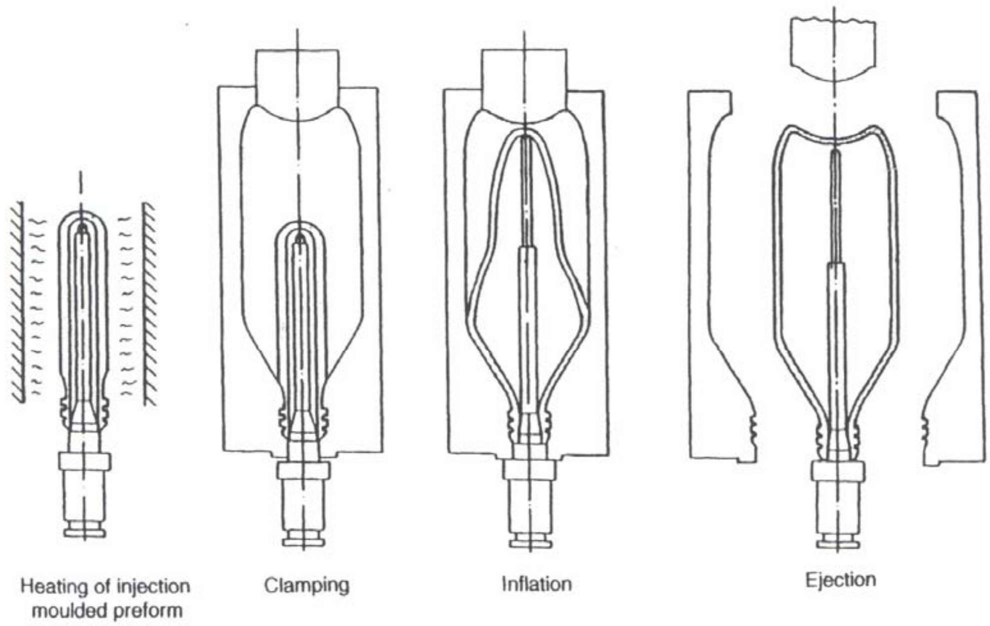


Fig. 17 Injection blow moulding process

The advantages of injection blow moulding are that:

(i) The injection moulded parison may have a carefully controlled wall thickness profile to ensure a uniform wall thickness in the inflated bottle.

(ii) It is possible to have intricate detail in the bottle neck.

(iii) There is no trimming or flash (compare with extrusion blow moulding).

A variation of this basic concept is the Injection Orientation Blow Moulding technique developed in the 1960s in the USA but upgraded for commercial use in the 1980s by AOKl in Japan. The principle is very similar to that described above and is illustrated in Fig. 18. It may be seen that the method essentially combines injection moulding, blow moulding and thermoforming to manufacture high quality containers.

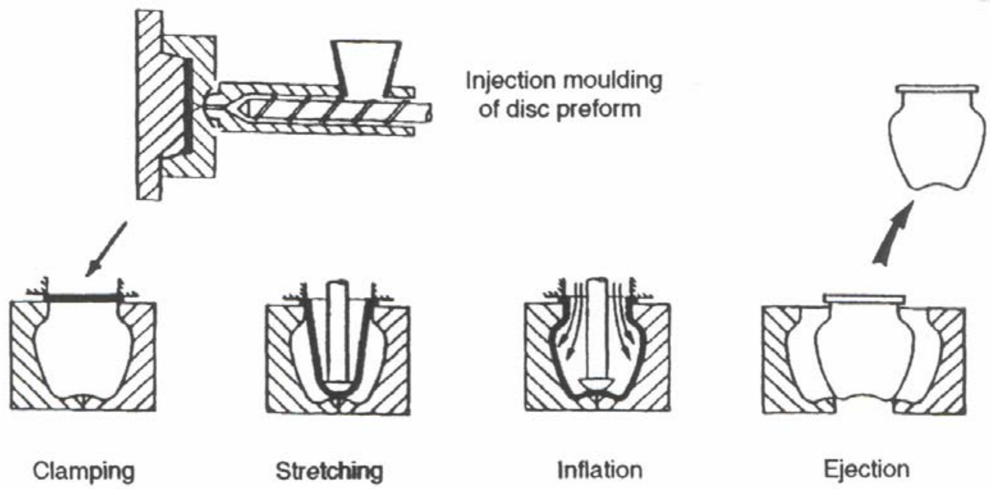


Fig. 18 **Injection** orientation strech blow moulding

Injection Moulding of Thermosetting Materials

In the past the thought of injection moulding thermosets was not very attractive. This was because early trials had shown that the feed-stock was not of a consistent quality which meant that continual alterations to the machine settings were necessary. Also, any undue delays could cause premature curing of the resin and consequent blockages in the system could be difficult to remove. However in recent years the processing characteristics of thermosets have been improved considerably so that injection moulding is likely to become one of the major production methods for these materials. The injection moulding of fiber reinforced thermosets is also becoming very common.

Nowadays, the injection moulder can be supplied with uniform quality granules which consist of partially polymerized resin, fillers and additives. The formulation of the material is such that it will flow easily in the barrel with a slow rate of polymerization. The curing is then completed rapidly in the mould.

In most respects the process is similar to the injection moulding of thermoplastics and the sequence of operations in a single cycle is as described earlier. For thermosets a special barrel and screw are used. The screw is of approximately constant depth over its whole length and there is no check value which might cause material blockages (see Fig. 19). The barrel is only kept warm (80-110°C) rather than very hot as with thermoplastics because the material must not cure in this section of the machine. Also, the increased viscosity of the thermosetting materials means that higher screw torques and injection pressures (up to 200 MN/m^2 are needed).

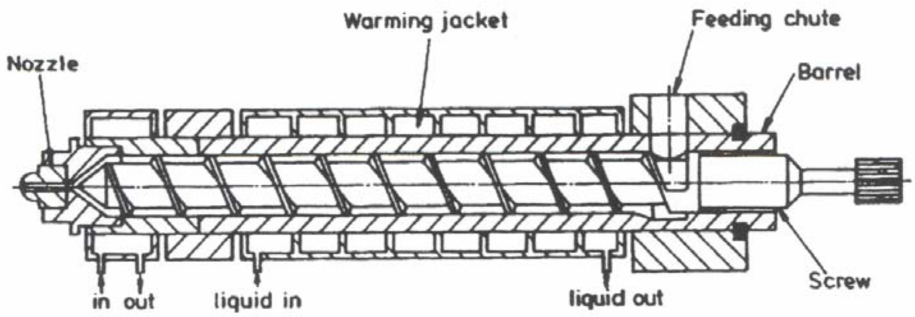


Fig. 19 Injection moulding of thermosets and rubbers

On the mould side of the machine the major difference is that the mould is maintained very hot (150- 200°C) rather than being cooled as is the case with thermoplastics. This is to accelerate the curing of the material once it has taken up the shape of the cavity. Another difference is that, as thermosetting materials are abrasive and require higher injection pressures, harder steels with extra wear resistance should be used for mould manufacture. As a result of the abrasive nature of the thermosets, hydraulic mould clamping is preferred to a toggle system because the inevitable dust from the moulding powder increases the wear in the linkages of the latter.

When moulding thermosetting articles, the problem of material wastage in sprues and runners is much more severe because these cannot be reused. It is desirable therefore to keep the sprue and runner sections of the mould cool so that these do not cure with the moulding. They can then be retained in the mould during the ejection stage and then injected into the cavity to form the next moulding. This is analogous to the hot runner system described earlier for thermoplastics.

The advantages of injection moulding thermosets are as follows:

(a) Fast cyclic times (see the Table)

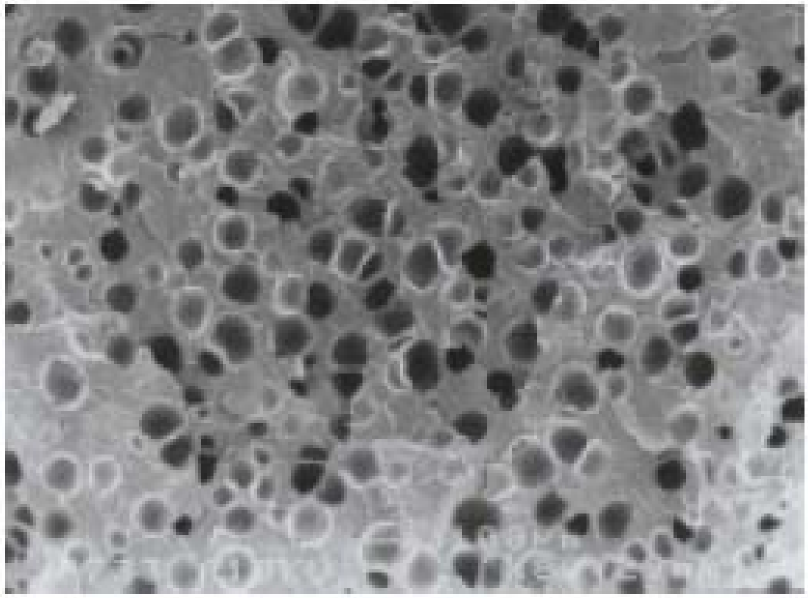
(b) Efficient metering of material.

(c) Efficient pre-heating of material.

(d) Thinner flash - easier finishing.

(e) Lower mould costs (fewer impressions).

	Compression Moulding	*Injection Moulding*
Open mould, unload piece	0.105	0.10
Mould cleaning	0.14	
Close machine, start pressure	0.10	
Mould cycle time	2.23	1.90
Total injection cycle (min)	2.575	2.00



Clamping Force Calculation

A) Clamping force for circular disc

The mould clamping force may be estimated in the following way:
Consider the moulding of a disc which is centre gated as shown in fig. 1 (a).

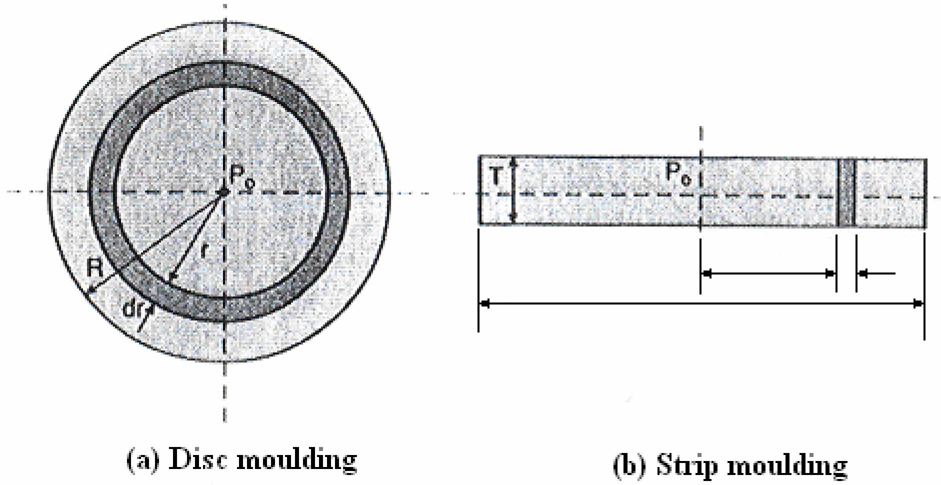


Fig. 1 Clamp force analysis

The force on the shaded element is given by:

$$Force\,, F = \int_0^R P_r \, 2\pi r dr$$

Experimental studies have suggested that an empirical relation of the form:

$$P_r = P_o\left(1 - \left(\frac{r}{R}\right)^m\right)$$

is most satisfactory.
P_o is the pressure at the gate.
m is a constant which is usually between 0.3 and 0.75 (pressure loss coefficient).

$$\therefore F = 2\pi \int_0^R P_o \left(1 - \left(\frac{r}{R}\right)^m\right) r dr$$
$$= 2\pi \int_0^R P_o \left(1 - \frac{r^{m+1}}{R^m}\right) dr$$
$$= 2\pi P_o \left[\frac{1}{2} r^2 - \frac{1}{m+2} \frac{r^{m+2}}{R^m}\right]_0^R$$
$$= 2\pi P_o \left[\frac{1}{2} R^2 - \frac{1}{m+2} \frac{R^{m+2}}{R^m}\right]$$
$$= \pi P_o R^2 \left(\frac{m}{m+2}\right)$$

This is a simple convenient expression for estimating the clamping force required for the disc.
An alternative way of looking at this equation is that the clamping pressure based on the projected area of the moulding, is given by:

$$\text{Clamping pressure} = \left(\frac{m}{m+2}\right) \times \text{injection pressure}$$

Where:

The material flow ratio $\frac{m}{m+2}$ may be determined from the flow curve.

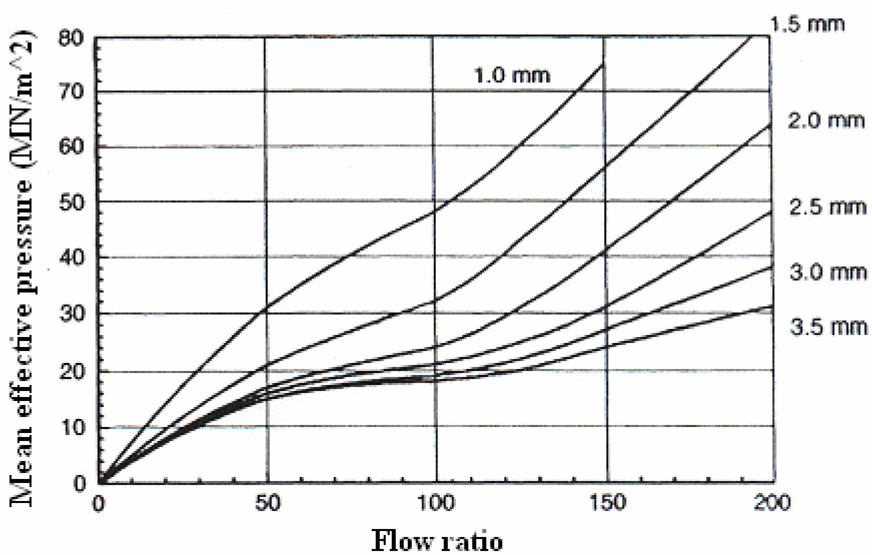


Fig. 2 Clamping pressure for different cavity geometries (typical values for easy flow materials)

Where:

$$\text{Flow ratio} = \frac{\text{Flow length}}{\text{Channel lateral dimension}}$$

$$\text{Clamping force} = \text{Mean effective pressure} \times \text{projected area}$$

- In practice it is necessary to increase this value by 10-20% due to the uncertainties associated with specific moulds.
- For plastics other than the easy flow (PE, PP, and PS) materials, it would be normal to apply a factor to allow for the higher viscosity.

Material	***Viscosity factor***
PP, PE, PS	1
Nylon 66	1.2-1.4
ABS	1.3-1.4
Acrylic	1.5-1.7
PVC	1.6-1.8
PC	1.7-2.0

B) Clamping force for rectangular strip

Consider the centre gated strip as shown in Fig. 1 (b):

$$Force\,, F = 2\int_0^{\frac{L}{2}} P_z T dz$$

Experimental studies have suggested that an empirical relation of the form:

$$P_r = P_o\left(1 - \left(\frac{Z}{L/2}\right)^m\right)$$

$$\therefore F = 2T\int_0^{L/2} P_o\left(1 - \left(\frac{Z}{L/2}\right)^m\right)dz$$

$$F = 2TP_o\int_0^{L/2}\left(1 - \frac{Z^m}{\left(L/2\right)^m}\right)dz$$

$$F = 2TP_o\left[Z - \frac{1}{m+1}\frac{Z^{m+1}}{\left(L/2\right)^m}\right]_0^{L/2}$$

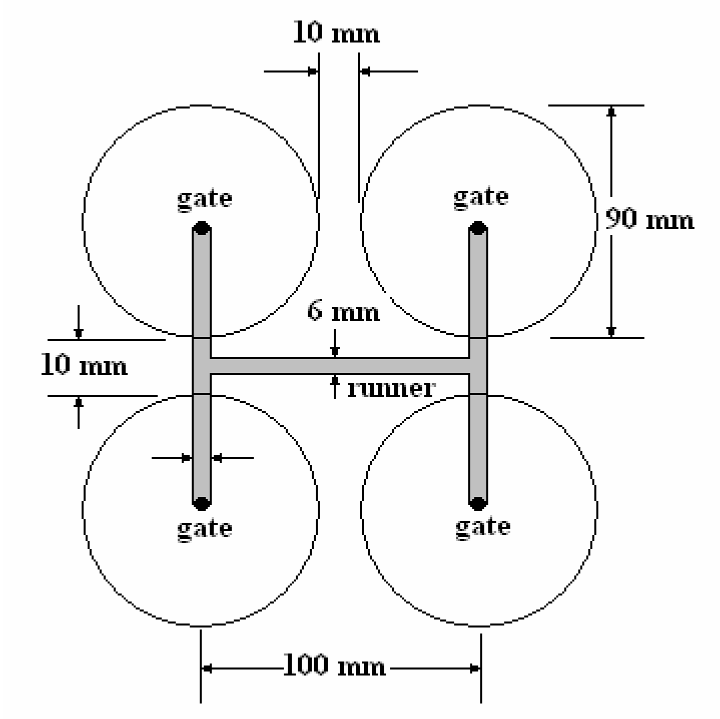
$$F = 2TP_o\left[\frac{L}{2} - \frac{1}{m+1}\frac{\left(L/2\right)^{m+1}}{\left(L/2\right)^{m}}\right]$$

$$F = TP_o L\left(\frac{m}{m+1}\right)$$

Example 1:

The mould shown produces four cup shaped ABS mouldings. The depth of the cups is 60 mm, the diameter at the base is 90 mm and the wall thickness is 1.0 mm. The distance from the sprue to the cavity is 100 mm and the runner diameter is 6 mm.

a- Calculate the clamping force necessary on the moulding machine if the mould is designed to feed cups through a pin gate in the centre of the base.
b- How could we minimize the clamping pressure?
c- Calculate the clamping force if the thickness of the cup increased to 1.5 mm.



Example 2:

A circular plate of diameter 0.5 m is to be moulded using a sprue gate in its centre. If the melt pressure is 50 MN/m^2 and the pressure loss coefficient is 0.6, estimate the clamping force required.