INJECTION MOULDING

THE INJECTION MOULDING PROCESS

The injection moulding process comprises three stages, each of which must be closely regulated to obtain good quality mouldings:

Feeding material into a heated cylinder, where it softens and becomes a plasticised melt;

Injecting the correct amount of plasticised material under controlled rate and pressure into an enclosed mould;

Maintaining sufficient pressure on the material to compensate for the shrinkage of the material on cooling as it cools to a point at which it can be ejected without deformation taking place.

The moulding machine often has interchangeable cylinders having varying shot capacities and different injection pressure maxima. The injection pressure and shot capacity are varied within the different cylinders by a change of screw diameter. The cylinders, which are generally coded A, B and C, change progressively through the range from smaller shot capacities at higher injection pressures to larger shot capacities at lower injection pressures. The most suitable cylinder for Lucite Diakon is the compromise B type. It is also good practice not to consider using more than 70% of the rated capacity of any given cylinder.

With high viscosity the injection pressures needed are correspondingly high and the mould must be of robust construction to resist these pressures and so prevent deformation under load. In addition the locking force, which keeps the mould closed during injection, must be adequate to resist the total thrust over the projected area of the mould cavity and so prevent the mould from opening. For this a minimum locking force of 30MPa of projected area should be available.

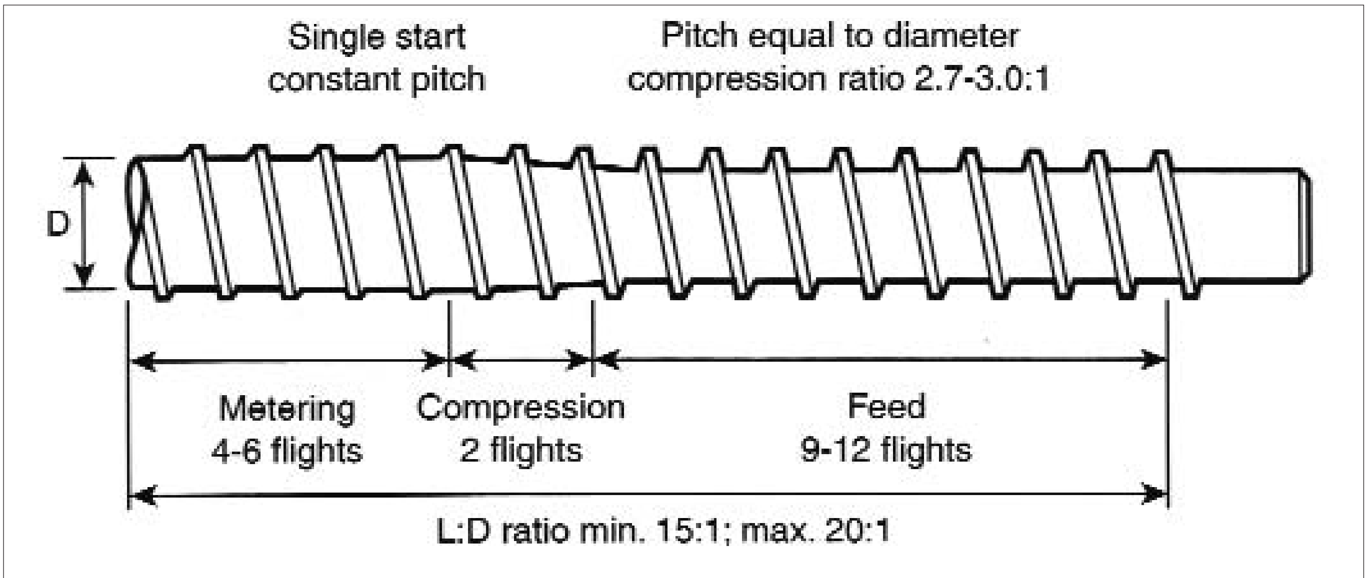


Figure 27 Screw Profile for Lucite Diakon

A stable and suitable rate of plasticisation is required to give a uniform and good quality melt for consistent shot to shot production of mouldings. See page 44 for moulding conditions.

The melt viscosity of acrylic is relatively high compared with, for example, polyolefine and polystyrene moulding materials, see Figure 47. Therefore the plasticising capability is important and the screw design in the majority of modern machines is adequate for processing Lucite Diakon. Figure 27 illustrates a suitable screw design for processing Lucite Diakon.

The quality of an injection moulded part is influenced by the temperature and pressure of material in the mould cavity at the moment when the material in the gate solidifies. At that instant the mould is filled with hot material under pressure. As

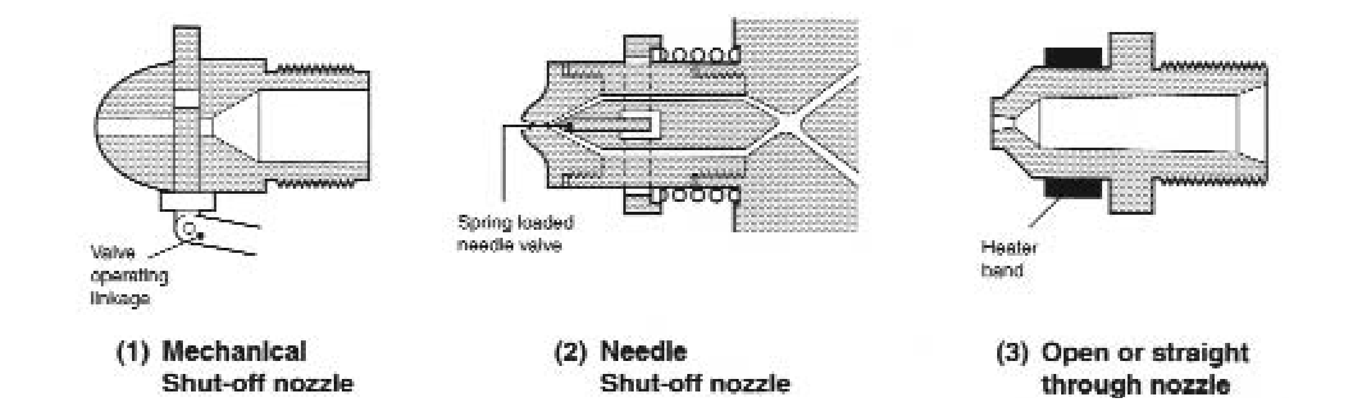


Figure 28 Nozzles

the temperature of the material in the mould falls there are two opposing actions taking place:

Thermal contraction - tending to reduce the volume of the moulding;
Residual pressure in the melt - tending to expand the moulding slightly.
The two effects occur at the same time and tend to counterbalance each other.

The use of programmed injection enables moulds to be filled at different speeds and pressure during the injection period. The advantage of being able to fill the major proportion of a mould quickly whilst at high pressure and speed and then drop to lower values maintaining follow-up pressure on the material helps to reduce the risk of flash and the degree of moulded-in strain. When using this system for thick acrylic sections, such as lenses and insignia, it is possible to inject very slowly at a low pressure and then, towards the end of the mould filling time, to increase the pressure to help overcome the shrinkage.

Control of mould temperature is also important if the quality of the moulded part is to be kept consistent throughout production. The use of mould temperature control units allows the mould temperature to be raised to its optimum value before start-up, thus avoiding an initial period of production of more highly strained parts from a cold mould, and wastage of material due to short shots.

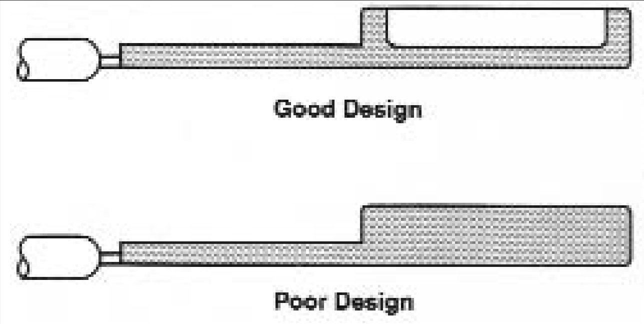


Figure 29 Design of components - change of section

On most of the screw pre-plasticising machines various types of nozzle can be fitted. Those nozzles fall into three basic categories
(see Figure 28).

(1) Mechanical shut-off nozzle
(2) Needle valve shut-off nozzle
(3) Open or straight though nozzle

For Lucite Diakon, nozzles (1) and (3) can be used quite successfully. However, nozzle (3) is usually preferred because there are no potential hold-up points where material can stagnate and decompose. This is very important when producing high quality clear mouldings. With this type of nozzle, however, close temperature control is required by means of a separate, well-positioned thermo-couple and temperature controller to prevent dribble or ‘freeze off’. Type (2) nozzles are not generally recommended because of the frictional heating which can occur when using high injection rates.

DESIGN OF COMPONENTS FOR MOULDING

Good component design is of great importance in the injection moulding of Lucite Diakon and the following points should be considered at the design stage if later difficulties are to be minimised or avoided.

If possible, sharp change of section thickness should be avoided as this creates excessive moulding flow problems with thicker sections and the possibility of excessive sinking on cooling if the gating position is only permissible at the thinner part of the moulding. To keep the cross-section constant, thick sections should be cored out wherever possible (Figure 29).

With certain designs, as for example the prismatic effect in a tap handle, thick and thin sections are closely alternated. In this instance the rapid change in section gives attractive optical results, but differential thicknesses must be kept within certain limits to avoid problems in moulding or in service. As a guide, the thickest sections should not exceed 10 mm. Even then, as the mould cavity fills, the melt will tend to flow into the thick sections first and the thin ones thereafter, leaving a weld line where the adjacent flows of melt re-unite. All edges of the core pin (which forms the hollow in the handle) should be radiused to ensure that the melt will not drag over them with consequent formation of flow lines, and to reduce the possibility of stress cracking by eliminating sharp corners in the moulding. Different cooling rates of thick and thin sections can also lead to stresses in the finished moulding.

To achieve economy, components are often reduced in section. This practice can be followed provided the sections are not made so thin as to cause flow problems during moulding. In addition to the flow problems, thin sections cool rapidly in the mould and result in high quenching stresses which make mouldings more liable to craze and crack. As a guide, where long flow paths are encountered, wall sections should not be less than 3 mm.

Problems resulting from uneven filling of the mould cavity will occur if the component is surrounded by a rim which is thicker than the internal portion in the centre (Figure 30). In this instance material will flow around the rim faster than across the centre and then give gas entrapment and “Y” weld line problems.

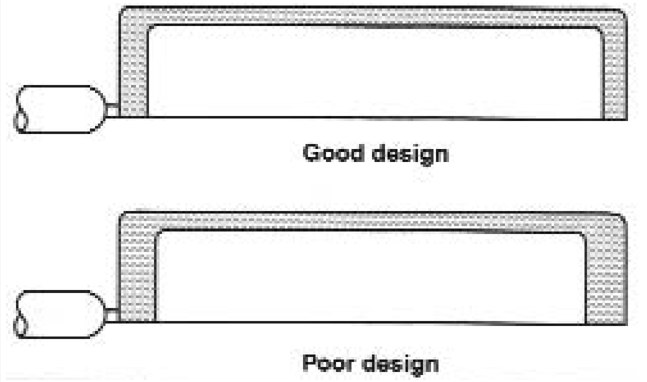


Figure 30 Design of components - thickness of rim

All corners and sections should be radiused as sharp corners cause stress concentration, brittle moulding and also pressure drop leading to possible problems in mould filling. This particularly applies where blind holes are to be moulded-in to take screw or other fastening media. Wherever possible all holes and slots should be moulded-in since post-moulding machining operations not only increase finished part cost, but also set up residual stresses which can only be removed by annealing.

To maintain the benefits arising from increased impact strength and flexibility with Lucite Diakon ST grades, it is important, at the initial design stage of components, to avoid sharp corners and sudden changes in section thickness, thereby eliminating areas subject to high tensile stress as with standard grades. All such corners should have a minimum radius of 1.5-2.0 mm. It is also important in the design and gating of the components to pay attention to the avoidance of weld lines as this effect, common to many impact modified plastics, is more noticeable than with a similar component in a standard Lucite Diakon grade.

If inserts are to be moulded-in, sufficient material should be allowed around the insert to give adequate keying and to resist stresses which will be set up during cooling by the differential thermal contraction of the insert and the Lucite Diakon. High softening point plastic inserts like glass reinforced nylon are preferable to metal to minimise the stress. The insert should be splined on the outside and provided with a circumferential slot to give a key to the Lucite Diakon which is moulded over it.

When numerals or letters are to be moulded into the component these should not be more than one third of the depth of the section thickness in order to minimise division of the melt leading to weld lines and 'tails'.

To aid ejection, the draft angle on a component should be as generous as possible. This especially applies to thick components where long injection times are often necessary and consequently increases moulding packing. In general 1° suffices for most thinner sections but as much as 3-4° may have to be accepted in extreme circumstances.

The shape of the component often dictates the positions of the mould parting line, gate and ejection points, and these should be taken into account at the design stage in order to facilitate the moulding of good quality components without objectionable appearance defects.

With tap handles or control knobs the use of a splined spigot is recommended.

With splined spigots the torque is distributed very evenly and a matching hole may therefore be moulded into the boss of the tap handle. The crests and valleys of the splining should be radiused to reduce and distribute any stresses which might be generated by excessive pressure.

With a square section spigot, the torque applied is concentrated at the internal angles of the moulded square hole, and cracking could occur.

MULTI-COLOURED MOULDINGS

The techniques described below have been highly successful with Lucite Diakon, particularly in the automotive industry on rear light assemblies.

Edge-to-Edge Insert Moulding

The process involves moulding part of a complete assembly in one tool, and transferring this part to a second tool, where further material is moulded against this insert. The hot melt fuses with the inserted moulding producing a bond between the two components. The strength of the bond is further increased if some form of mechanical key is designed into the joint area. The design of this key depends largely on the shape of the component. A few examples are given in Figure 31.

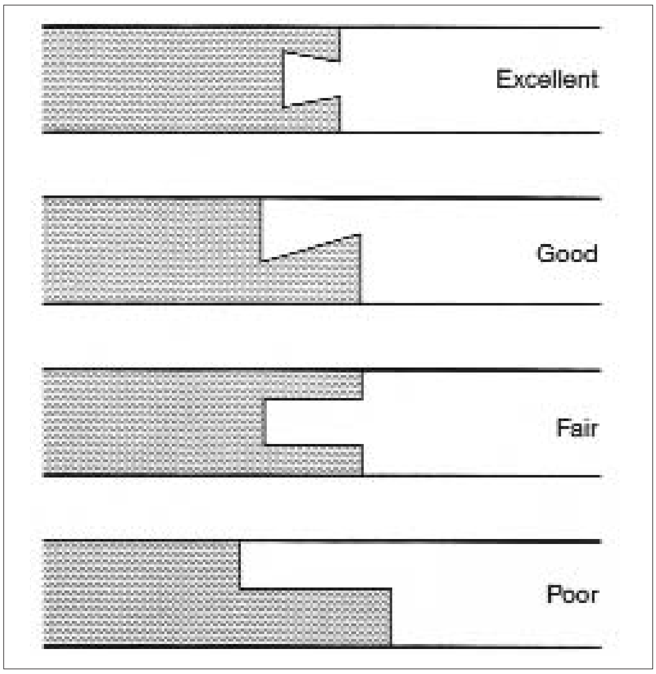


Figure 31 Forms of key with edge-to-edge moulding

The second tool must be accurately designed to accept the inserted moulding which must be held and supported firmly during the moulding cycle. Accuracy in both component and mould is also required to prevent flashing between the two components.

The pressures exerted upon the inserted moulding during the second moulding cycle are often high and, to avoid cracking, components must be designed without sharp corners. The gate should be positioned to minimise the strain on the inserted moulding. When moulding Lucite Diakon, it has been found advantageous to use the higher molecular weight grade CMH454V or low impact versions of ST for the moulding to be used as the insert, using their superior mechanical strengths to help prevent cracking.

Ideally the two moulding operations should be carried out consecutively on adjacent machines, the moulded inserts transferred directly from one machine to the next. Under these conditions the inserts are still warm and the risk of cracking is reduced. If direct transfer is not possible then it is helpful to warm the mouldings prior to placing them in the second tool.

Skin Insert Moulding
This process is similar to that described above except that the insert is a thin component with smooth surfaces, normally in the region of 1.5 mm thick. Many of the points mentioned above apply to skin moulding. The final part is produced by moulding a second layer skin, which will include the optics, on to the first in a master tool. It has been found that when required the use of CMH454V for the insert skin helps to minimise cracking or colour bleeding problems. Although, due to component and tooling considerations, both edge-to-edge and skin moulding techniques are used, it is considered skin moulded lenses are more robust than edge to edge ones.

Multi-Colour Machines
Multi-colour moulding may also be carried out on special machines with two or more cylinders for those rearlight assemblies where design, size and number considerations are suitable. The technique usually consists of a series of moulds where one platen is rotated through two or more stations where injection of the different coloured material takes place.

MOULD DESIGN

Although many factors have to be considered in the design of moulds for thermoplastic materials there are three factors which require special attention for acrylic materials. Due to the relatively high melt viscosity and its greater temperature dependence (see Figure 47) it is usually necessary to use sprues, runners and gates of generous cross section compared to those used for material with low viscosities such as nylon and polystyrene. Standard grades of Lucite Diakon may be considered as hard brittle materials and allowance should be made for this.

- **Radius all corners**
- **Adequate and uniform ejection**
- **No undercuts**
- **Minimum 1° taper**
- **Polish in line of draw**
- **Uniform mould temperature control**

The aesthetic appearance together with high gloss and clarity obtainable with Lucite Diakon mouldings requires highly polished moulds.

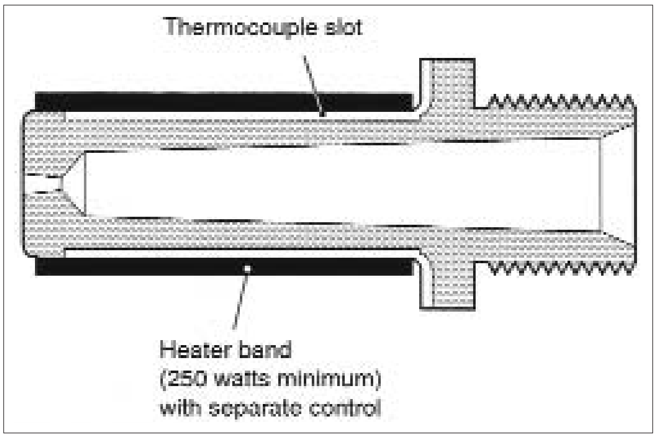


Figure 32 Extended nozzle

In general nickel-chrome steels are preferred since in addition to being tough and hard-wearing they will take a high polish. For optical quality parts, a steel like 'Stavex'* ESR has been found satisfactory.

Sprue Design
The sprue is the channel through which the material is transferred from the machine nozzle to the runner(s) and gate(s) and into the mould cavity (ies). Its design, therefore, is of paramount importance. It must be of adequate dimensions to prevent freezing prematurely, but not so large as to extend the cooling time of the moulding. To fulfil these basic requirements it is thus important to have a sprue of adequate diameter but to keep it as short as possible. A length of approximately 60 mm should be aimed for. To achieve this mould backing plates or bolsters should be kept as thin as possible without sacrificing strength in the mould.

Where it is not possible to provide short sprues due to component geometry, consideration should be given to the use of extended machine nozzles (see Figure 32) which can be fitted with suitable heaters and controlling equipment. An extended nozzle can also be used to advantage on normal type moulds in order to obtain better mould filling and to reduce material wastage.

The size of the sprue necessary for any particular moulding will vary according to the thickness and the shape of the parts to be made.

* Trademark of Uddeholm

As a guide the following sprue diameters should be used:

For thin section moulding, ie 2.5 mm-4 mm, the machine nozzle should be 4 mm diameter and the smaller hole in the sprue bush 4.5 mm diameter; For thick section mouldings, ie 6 mm and upwards, the machine nozzle should be 7.5 mm diameter and the sprue accordingly 8.5 mm diameter.

All sprue bushes should have a tapered bore to allow easy extraction of the sprue. The angle of the taper should be between 5-7° inclusive. The higher angle is preferred for thick mouldings because the long injection times necessary for these mouldings can cause packing which tends to make the sprue more difficult to extract. The sprue bush internal surface should be free from machine and grinding marks and should preferably be draw-polished. A generous 'cold slug-well' should be positioned opposite the entrance of the sprue into the mould whenever possible. In addition to removing the piece of slightly chilled material left in the nozzle from the previous shot, it may also be designed with a 'Z pin or back taper to aid the removal of the sprue from the sprue-bush. The cold 'slug-well' is ejected with the moulding and runner system.

Runner Design

To facilitate the production of good quality mouldings, particular attention should be paid to the design and layout of the runner system.

Runners, like sprues, should be generous in diameter and short in length to minimise pressure loss and permit adequate follow-up pressure in the initial stage of cooling.

Full-round runners give the best results (see Figure 33) but if these cannot be used trapezoidal runners can be used satisfactorily. Half-round and flat runners tend to cause premature freezing of the melt and should not be used. In multi-cavity moulds it is necessary to balance the runner layout by having main and secondary runners to achieve even pressure transmission into each cavity of the mould. A cold slug overflow well should be provided at the end of main runners.

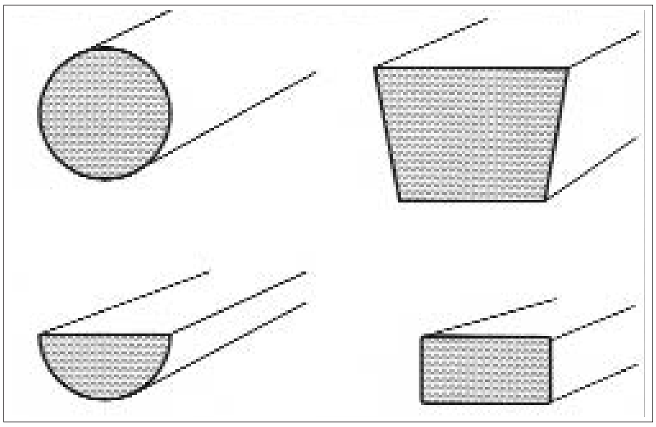


Figure 33 Runners

As a guide to runner design and size the following should be used:

For thin-section mouldings, ie 2.5 mm-4mm, the main runner should be 6 to 8 mm in diameter.

For thick-section mouldings, ie 6 mm and upwards, the main runners should be 10 mm and above.

The large diameter runners are usually necessary for items such as lenses, brush backs, insignias, etc. If secondary runners are to be used they should not be significantly smaller than the main runner.

Hot Runner Moulds

The use of the hot-runner technique for feeding multi-impression and large area mouldings is now firmly established in the acrylic moulding industry. The advantages of hot-runner mouldings are:

Melt enters the cavities in a more controlled condition than with a sprue and runner system, as temperature control in the hot runner is adjustable to finer limits;

A possible reduction in post-moulding finishing operations to remove large sprue gate witness marks;

The elimination of cold sprues and runners in multi-impression moulds which would normally be scrapped or reworked;

Hot-runners enable single impression, large area mouldings to be edge-gated, whilst keeping the moulding in the centre of the machine platen.

Effective increase in the shot capacity of the machine as, once the hot-runner is filled, the injection capacity can be fully concentrated into the cavities.

In designing hot-runner moulds (Figure 34) the following important points should be observed:

Provide adequate heating for the hot runner manifold (1.8 watts/cm^3 or 30 watts/in^3) and nozzle (approximately 300 watts);

Make provision for closely controlling the temperature of the manifold and nozzles with suitable instruments;

Insulate the hot-runner manifold and nozzles from the machine platen or mould cavities by air or compressed temperature-resistant sheeting;

Provide adequate runner channels in the heated manifold, ie minimum 12 mm diameter;

Make the machine nozzle orifice diameter of similar size to the channels in the hot-runner manifold;

Ensure that the runner channels are devoid of any sharp corners or blind spots where melt could become trapped and consequently degraded.

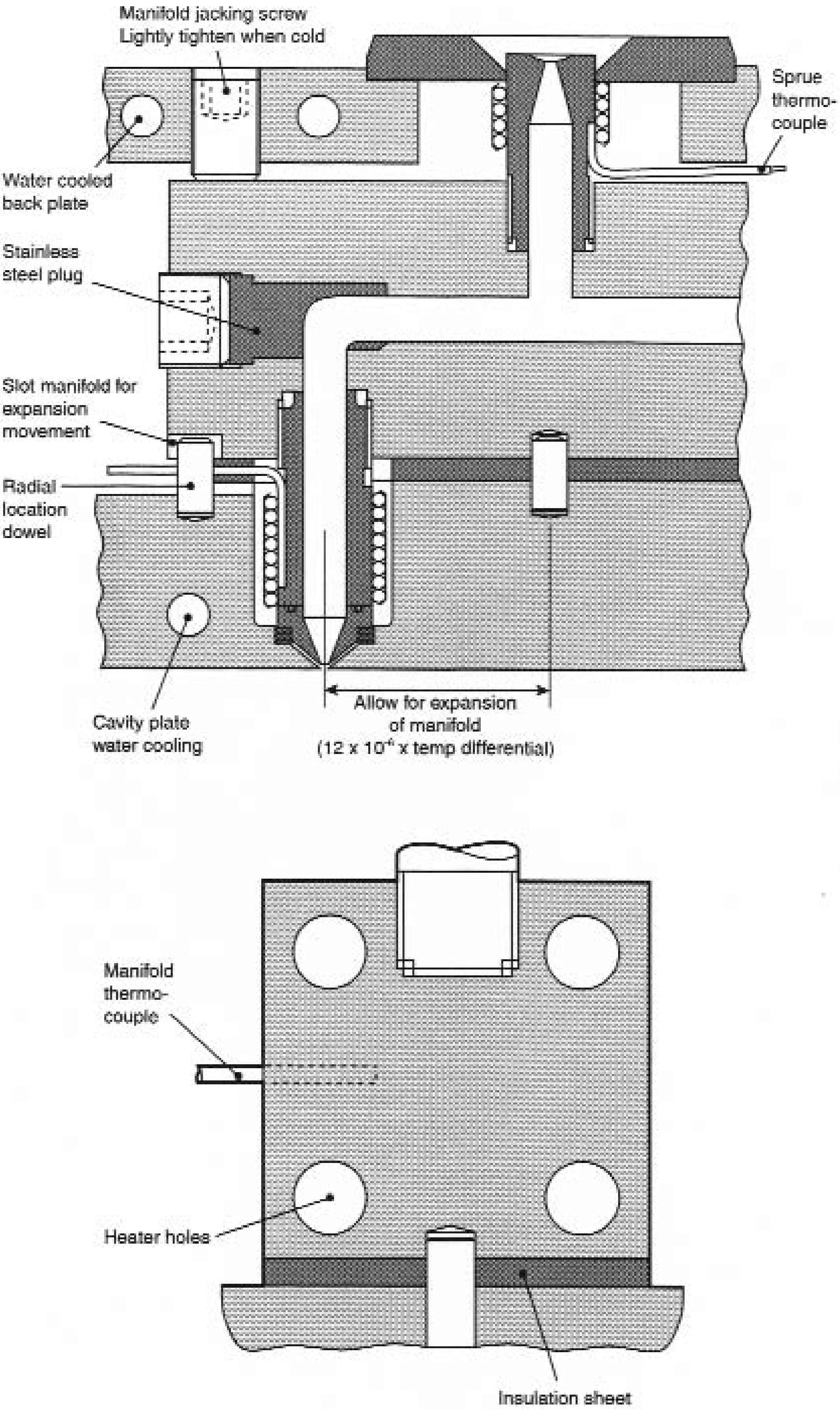


Figure 34 General Assembly of Hot Runner Mould

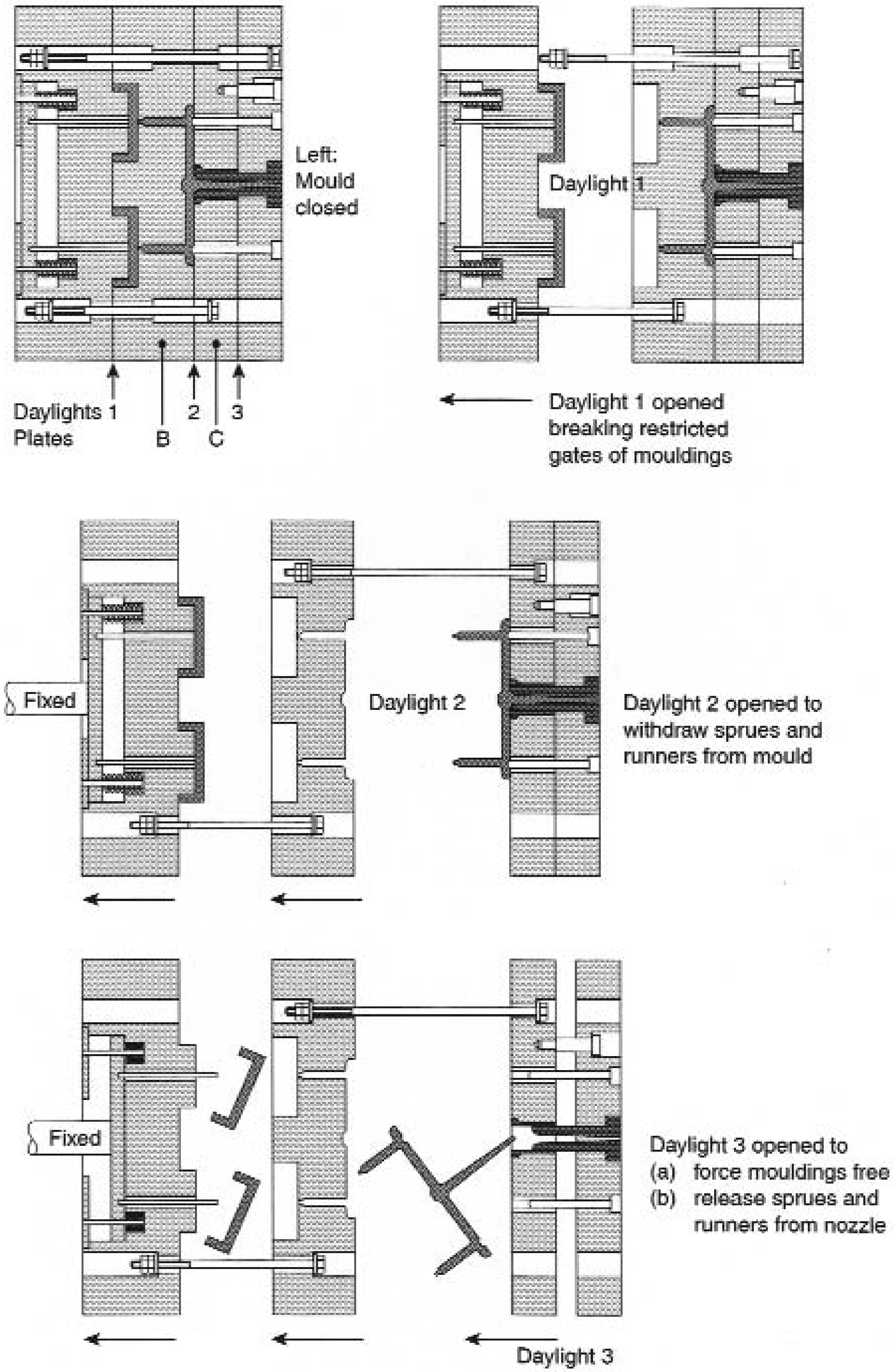


Figure 35 General assembly and operation of typical three-plate mould

Three-Plate Moulds
These are normally used when multi-cavities for small components are involved and semi- or fully-automatic working is required. However, as indicated earlier, due to the brittle nature of acrylic materials, this type of design has to be used with care.

This type of mould, as it name suggests, has an extra plate (see Figure 35). This plate (B) usually carries on one side the gate and the complete runner system, preferably trapezoidal, and on the opposite side the plate carries part of the mould form (usually the female part).

When the mould opens plate (B) is separated by means of a delayed action mechanism (eg chains or length bolts), so breaking the restricted gate. The mouldings are then ejected from one daylight and the sprue and runner system are ejected from the other.

Successful ejection of mouldings relies on clean separation of the moulding and gate at the parting line.

With this method of tooling, restricted gates of the correct design must be used (see Restricted gate, Figure 37).

Multi-plate moulds are usually more expensive than two-plate moulds and can be slower in production if an operator has to remove the sprue and runner system when the mould is open. This can usually be avoided by providing automatic ejection of sprue and runner. Such a mould is shown in Figure 35 where in addition to plate (B) the runner is stripped out automatically with a runner stripper plate (C). The distance travelled by the plates is governed by the length of the chain or the length of the bolts used to separate them.

Gate Design
The type and position of the gate is often dictated by the design of the component and the number of mouldings to be produced in each cycle. For guidance the following section provides information on different gating methods.

Sprue Gate (Figure 36)
This type of gate is the preferred gate and is normally used for single-impression moulds, especially suitable when the component is cup shaped and involves a base. Its advantage over a side gate is that the flow ratio is reduced and the mould will be filled symmetrically. This system may be extended to multi-impression moulds in conjunction with a hot-runner assembly.

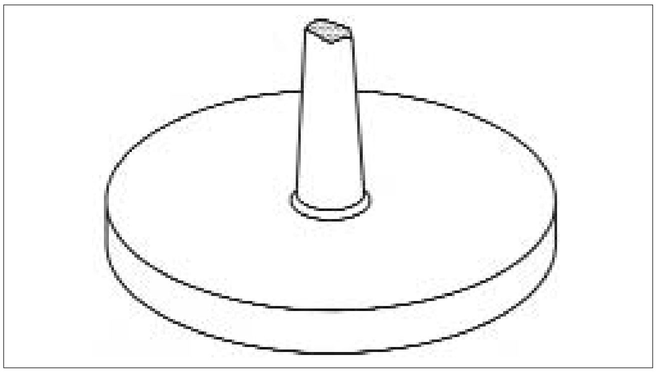


Figure 36 Sprue Gate

Restricted Gate (Figure 37)
This type of gating is used for multi-cavity tools. Finishing operations can often be eliminated because the small gate is broken off during the ejection of the moulding. The gate must not be too small otherwise the filling of the cavity is impaired. Also, under the effect of high injection pressures frictional heating of the material passing through the gate could lead to splash marking and burning on the finished component. However the gate must not be made too large otherwise it will not break off satisfactorily during ejection. As a guide restricted gates should not be smaller in diameter than 1.0 mm or greater than 1.8 mm. It is also essential to have a generous runner system to prevent premature freezing of the melt.

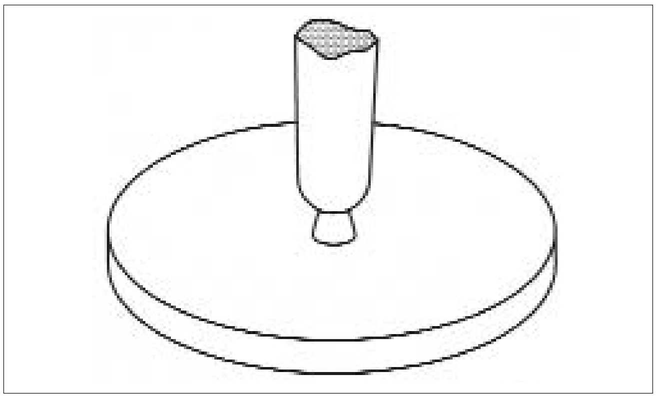


Figure 37 Restricted Gate

To prevent any cracking around the gate during the ejection of the moulding (particularly where larger gates are being used) the gate should have a slight back taper so that it breaks off about 1.5 mm from the surface of the moulding.

Owing to the notch effect, restricted gates should be located at a point in the moulding subject to low mechanical stresses. Also, where a clean finish is required, the pronounced orientation of the material in the gate area often hinders the removal of the gate-mark by milling, due to small cracks occurring along the lines of orientation. Hence care should be taken in the removal of any restricted gates.

Side or Edge Gate (Figure 38)
This is the most common type of gate used to produce components of a flat or shallow nature. The size of the gate is dependent upon the shape and thickness of the moulding. For normal 2 to 4 mm thick mouldings the gate thickness should be two thirds that of the moulding. For thicker sections the gate thickness should be approximately 75% of the component thickness and as wide as the runner. With multi-cavity moulds where the gates are arranged in series, it is necessary to balance the filling of the cavities. This is not always easy to predict at the design stage of a mould and it may be necessary to complete the balancing operations by trial runs. Generally the gates furthest from the sprue are given the greatest cross-section and those nearest the sprue the smallest.

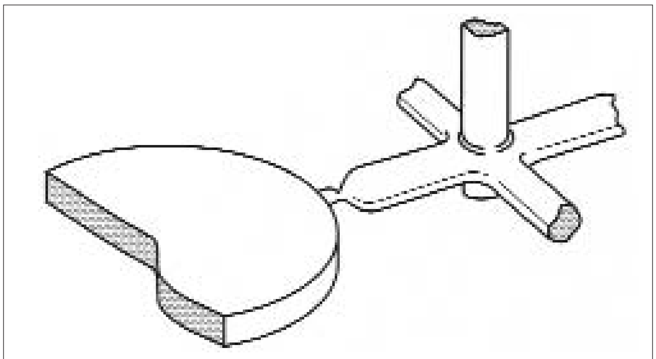


Figure 38 Side or Edge Gate

Flash Gate (Figure 39)
For long flat components of thin section this type of gate can be used quite successfully. It enables a large cavity to be filled quickly and consistently. The length of the gate is dictated by the length and width of the article and the flow pattern required. In some instances it is advantageous to have the gate the full length of the article, though usually a gate length which is about 50% of the longer side dimension is sufficient. However, it is important to retain adequate thickness of the gate and therefore more complex finishing operations will be required.

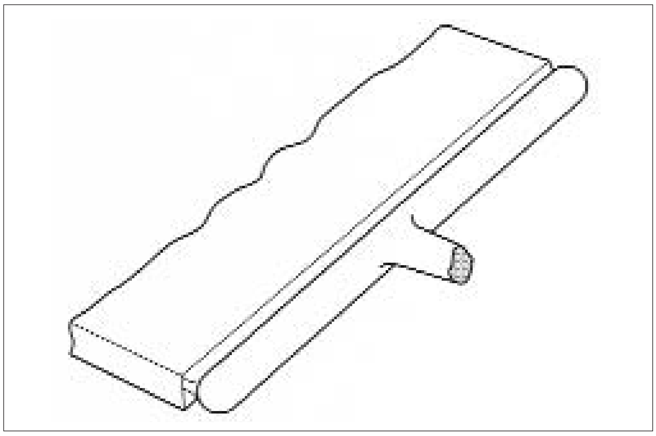


Figure 39 Flash Gate

Fan Gate (Figure 40)
For thick section mouldings such as optical lenses, this type of gating is used. It enables the runner to be made of adequate size to aid flow and prevent the material from chilling off when it is injected slowly as is necessary when making these components. It also allows sufficient follow-up pressure to be maintained on the cavity during the cooling contraction stage.

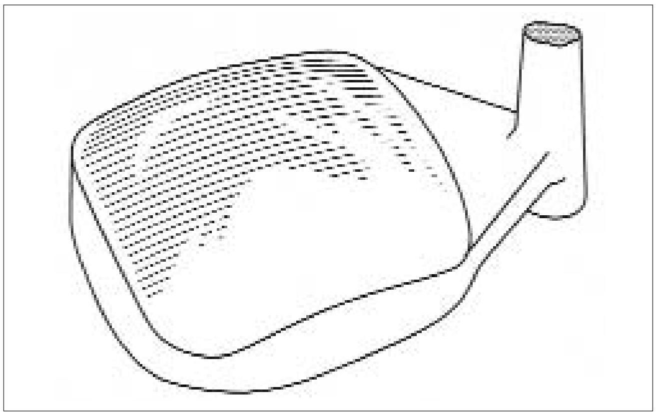


Figure 40 Fan Gate

Tab Gate (Figure 41)
This type of gating can be used as an alternative to side gating to produce articles of a flat or shallow nature. It has certain advantages over normal side gates in that the design minimises the jetting of material into the mould cavity which may lead to weld lines and flow marks.

Tab gates are normally used to produce elongated articles such as radio scales and rules. The tab in these instances is located towards one end so that the mould cavity is filled evenly down the greater part of its length. The longitudinal orientation of the material tends to strengthen the article and, because the gate is remote from the centre point of maximum stress, it avoids the risk of cracks developing at the gate area if the moulding is subsequently flexed.

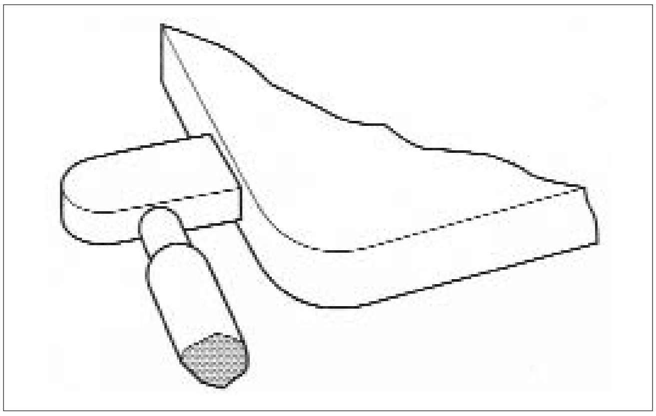


Figure 41 Tab Gate

Diaphragm Gate (Figure 42)
For single-impression moulds which are to be produced with a central orifice, this type of gating can be used to obtain uniform radial mould filling. The diaphragm gate is removed by a subsequent machining operation.

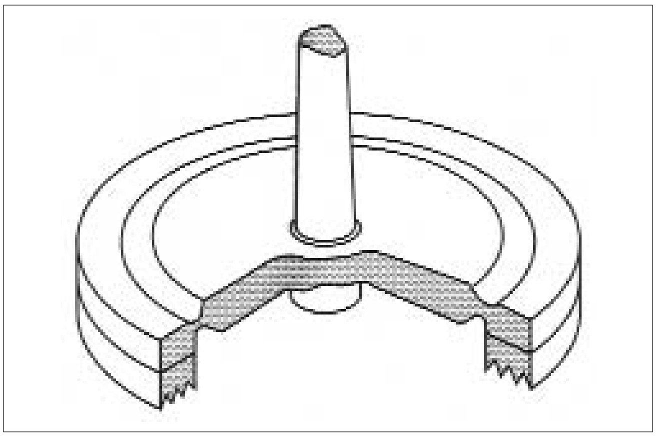


Figure 42 Diaphragm Gate

Spider Gate (Figure 43)
This is a variation of the diaphragm gate. It is normally used for moulding large diameter apertures and helps to reduce material wastage. A disadvantage is that weld lines are created by the meeting of the separate flow streams and this factor needs to be considered at the component and mould design stages.

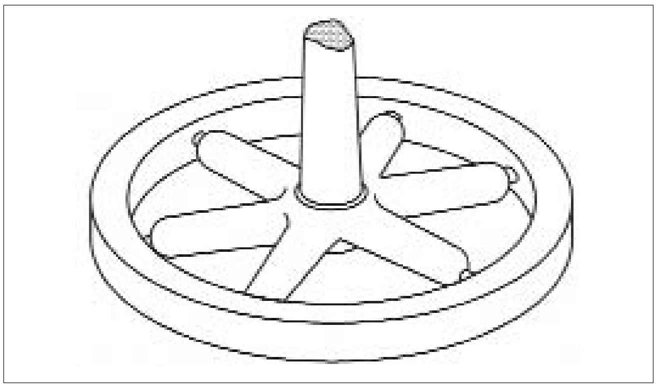


Figure 43 Spider Gate

Ring Gate (Figures 44 and 45)
For single or multi-impression moulds which are to produce tubular type articles this type of gate ensures consistent filling of the moulds. It also helps to ensure that the core pin is central with the cavity, whereas using an ordinary side gate the initial pressure would tend to displace the core pin and so cause the article to have an uneven wall section.

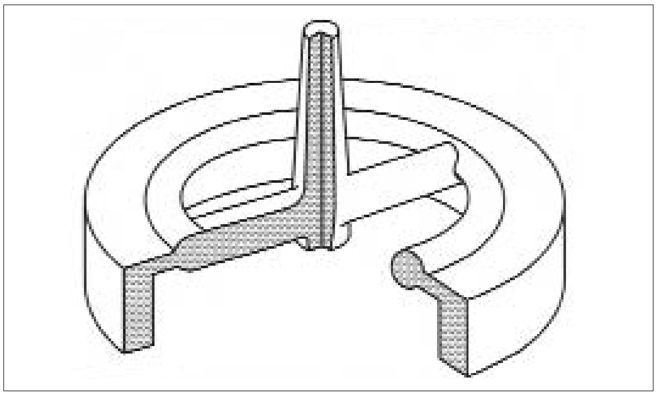


Figure 44

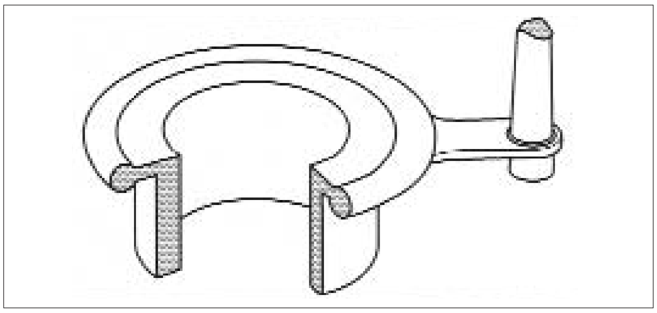


Figure 45

Submarine (Tunnel) Gate (Figure 46)
Although not recommended, this type of gate can be used on multi-cavity moulds in a similar manner to the restricted gate. It is normally used for articles which cannot have a mould mark on the base or for a tubular type article. The submarine gate differs from the restricted system in that is below the parting line of the mould. This means that the gate will not break off until the moulding is ejected. It is essential when using submarine gates to have a sufficient taper on the gating system so that the portion below the split line of the mould can be easily removed with the runner system. This system can be used to advantage with fully automated moulds.

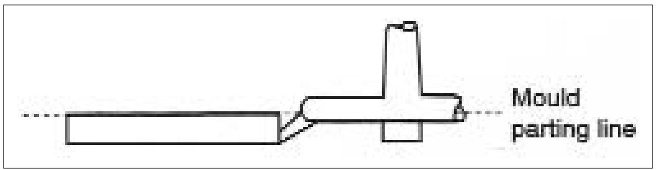


Figure 46

Gating of Thick Sections
To prevent sink marks and voids which must be absent when moulding lenses and prisms, the material shrinkage (a few per cent from melt to solid state) must be compensated by the flow of additional material into the mould during cooling. This flow of material can occupy several minutes depending on the thickness of the moulding. Hence the cross-section of the runner and gate must be of adequate size to prevent the gate freezing-off too soon.

The edges of the components must not be too thin, as could occur with the edge of a lens, since insufficient area would be available for the gate. In producing thick section articles of this type the gate thickness is more important than the width and should, in general, be at least three-quarters the thickness of the edge section. In order to prevent any flow lines the edge of the gate should be slightly radiused and the cavity must be filled slowly. Runner lengths should be kept to a minimum.

MOULDING TECHNIQUE

Care of Raw Material
Lucite Diakon acrylic polymers are normally suitable for moulding without any preliminary drying operation. However, moisture will be absorbed if the material is exposed to the atmosphere for excessive periods, or if material is kept under damp storage conditions. Material should not be allowed to remain in machine hoppers for more than a few hours. When not in use, bags and containers should be sealed and re-used as soon as possible. Stock control should be practised so that material storage time is kept to a minimum (recommended maximum 3 months) and the risk of moisture pick-up, through prolonged storage, reduced.

If material has become wet because of incorrect storage or handling, splash or mica marks will be observed on the surface of the moulded article. For best results wet material should be dried with dehumidified air driers at a temperature of 80-90°C for the Lucite Diakon M grades (eg CMG) [type 8] and 65-75° for the Lucite Diakon L grades (eg CLG) [type 6] with the residence time in the drier not less than 4 hours. Temperatures at or below the minimum will require longer in the drier while excessive temperature may lead to sintering of the granules. If the throughput of the moulding machine is greater than the time capacity of the drier problems may occur if the moisture is not only at the surface but has to diffuse from the centre of the granule.

In those more critical moulding applications it may be advantageous to pre-dry or pre-heat the material straight from the bag or container prior to moulding.

Rework Material
There is a tendency for the original water-white colour of acrylics to deteriorate slightly with repeated reworking and hence it is recommended that the amount of added rework material (scrap moulding, sprues etc ground up for re-use) should be limited when the moulded colour is critical. For applications where colour is less critical, a common addition level is 20%. Up to 100% of good quality rework may be used with no significant fall-off in properties but it is not to be recommended.

It is essential to ensure that the grinder is clean and that dirt contamination is not included during the grinding process.

The screen size on the grinder should be 3 mm- 6 mm. Larger screens should not be used since difficulty could be encountered in feeding, melting and processing larger particles, particularly if rework material is being blended with coloured material or used on shallow-flighted screws.

It is usually necessary to dry rework material prior to moulding if it has been exposed to the atmosphere for any length of time. The drying conditions for rework material are the same as used for virgin material. It is generally possible, by grinding sprues and runners soon after they have been moulded and keeping the material protected from the atmosphere, to mould it without drying.

Contamination

Lucite Diakon is not compatible with other moulding materials and strict precautions must be taken to prevent contamination which is immediately visible because of the high transparency of the material.

Contamination with other clear materials (polystyrene and polycarbonate) results in white cloudy streaks due to differences in refractive index.

Because Lucite Diakon is a good electrical insulator, it will pick up atmospheric dust by electrostatic attraction. Care must therefor be taken when loading machine hoppers to prevent unnecessary exposure.

Purging

Being a clear material, the changeover from other materials to Lucite Diakon is more difficult than with opaque plastics, and many moulders keep a separate cylinder soley for moulding acrylic. Where a separate cylinder for acrylic is not available the most convenient way to clean the cylinder, apart from a complete strip down, is to purge the machine using rework Lucite Diakon with the nozzle removed. The nozzle can be ‘burnt out’ separately.

Where black or heavily filled materials are to be removed from the cylinder it is useful to use scrap natural unfilled polypropylene as a purging compound before changing over to rework Lucite Diakon.

When purging it is recommended that the cylinder temperatures be raised during the initial stages of the operation. This assists removal of material from cylinder walls. Obviously care must be taken not to disrupt the carbonised layer on the screw and barrel or use excessive temperatures which could cause severe decomposition of the material. After a short while, temperatures should be reduced and the machine purged with Lucite Diakon at lower temperatures to remove remaining traces of unwanted material. Once the purging operation is complete a clean nozzle should be fitted.

Temperature Control

The melt viscosity of acrylic is more temperature dependent than that of many other thermoplastic materials as can be seen in Figures 47,48 and 49. It follows, therefore, that the moulding conditions must be accurately controlled.

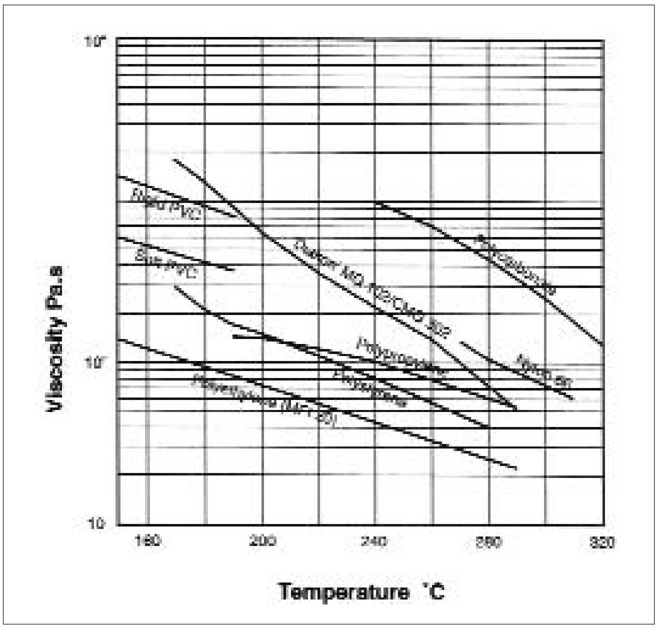


Figure 47 Variation of melt viscosity with temperature for different thermoplastics (Shear rate 1000 s^{-1})

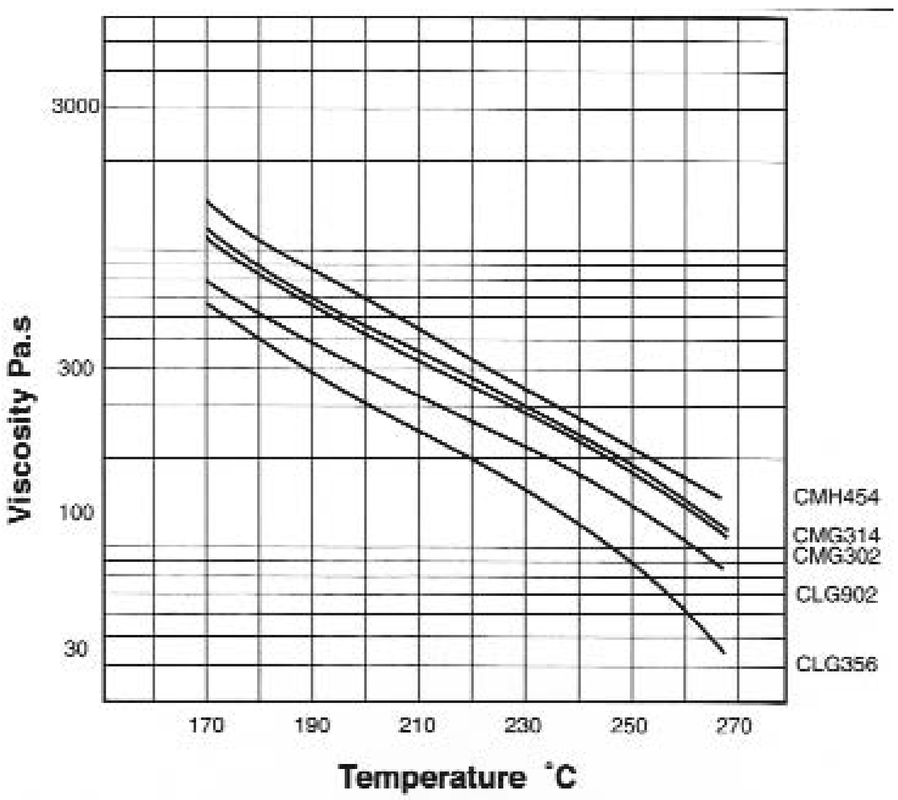


Figure 48 Variation of melt viscosity with temperature for different grades of standard Lucite Diakon (shear rate 1000s^{-1})

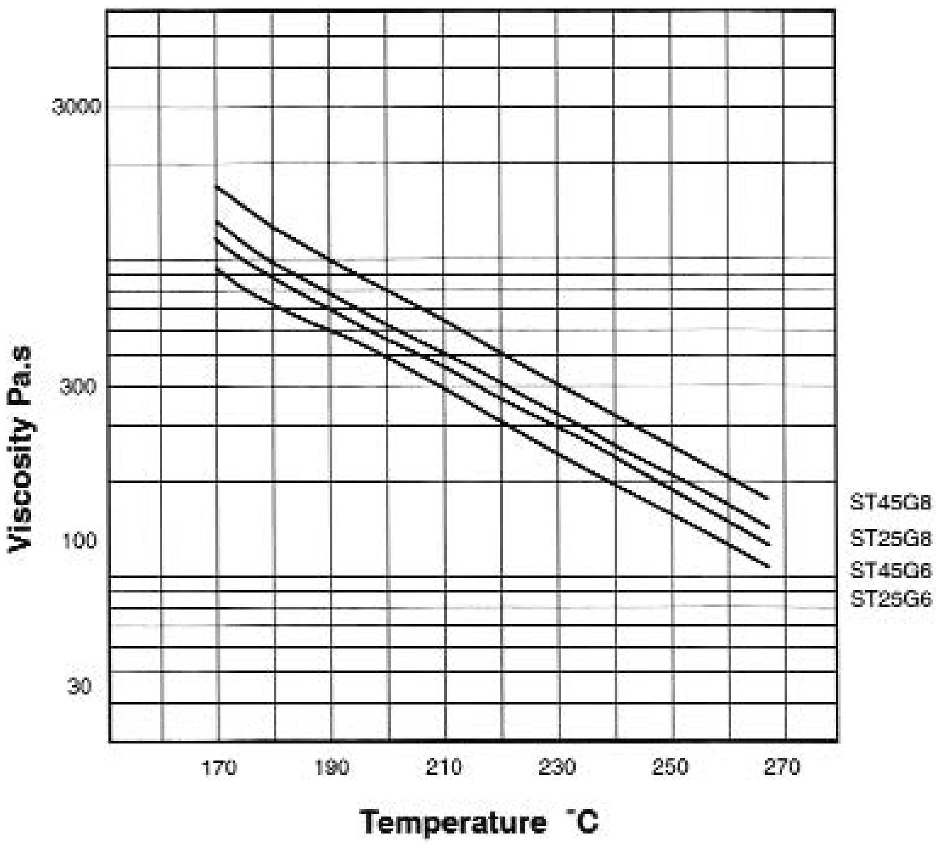


Figure 49 Variation of melt viscosity with temperature for different grades of Lucite Diakon ST (Shear rate 1000 s^{-1})

Position of Thermocouples

Controlling thermocouples should be located as close as possible to the heaters they control, eg in a slot directly under the band heaters. This arrangement eliminates any time lag in the response of the controllers and minimises cyclic variations in temperature.

Measurement of cylinder wall temperatures may be made by a set of deeply recessed thermocouples connected to a separate recorder. Such facility is not essential for production purposes but it is a useful guide for establishing optimum conditions and for experimental work.

Nozzle Temperature Control

This subject is discussed on page 30 but it is recommended that wherever possible separate control of the nozzle temperature should be used.

For long or extended nozzles separate control is essential to minimise any defects such as matt patches or splash marking around the sprue which occur because of the nozzle being too cold or too hot.

Moulding Conditions

The actual moulding temperatures and pressure setting required will vary from grade to grade and from one type of machine to another, depending on the size of the machine and the shot weight of the moulding. They will also depend on the design and section thickness of the component. The material temperature may be higher or lower than the indicated cylinder temperature depending on the amount of frictional heat introduced by the screw. It is therefore not possible to be specific about the exact moulding conditions for Lucite Diakon and each case must be considered on its own merit and in the light of experience.

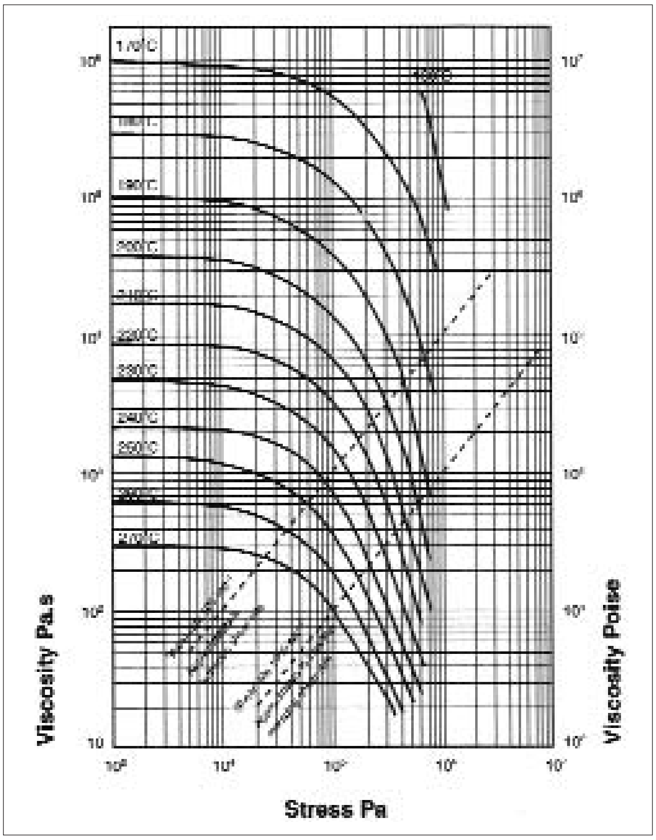


Figure 50 Variation of melt viscosity with stress for CMG302 at different temperatures.

However, remembering that Lucite Diakon has a high melt viscosity which is very temperature dependent when compared to many other thermoplastic materials, the moulding conditions in table 6 may be used as a guide for all grades, using the higher end of the melt temperature range for higher viscosity grades.

	Moulding Type		
	Normal	**Large Area**	**Thick Section**
Melt	230 to 250°C	260 to 270°C	As low as 180°C
Mould temperature	60 to 70°C	70°C	70°C
Screw speed	Medium	Medium	Slow
Back pressure	Low (to medium)	Low (to medium)	High (to medium)
Injection speed	Medium to fast	Medium to fast	Slow to very slow
Cycle time	40 seconds	70 seconds	> 2 minutes

Table 6 Guide to moulding conditions

Gate Size

The influence of gate size on mouldability or flow ratio of Lucite Diakon cannot be overstressed. There is a natural desire to use small gates to minimise both finishing operations and gate witness marks. However, the quality and ease of producing mouldings are significantly improved by using large gates with a balanced sprue and runner system. Figure 51 shows the influence of gate size and melt temperature on the approximate flow ratio for Lucite Diakon CMG302.

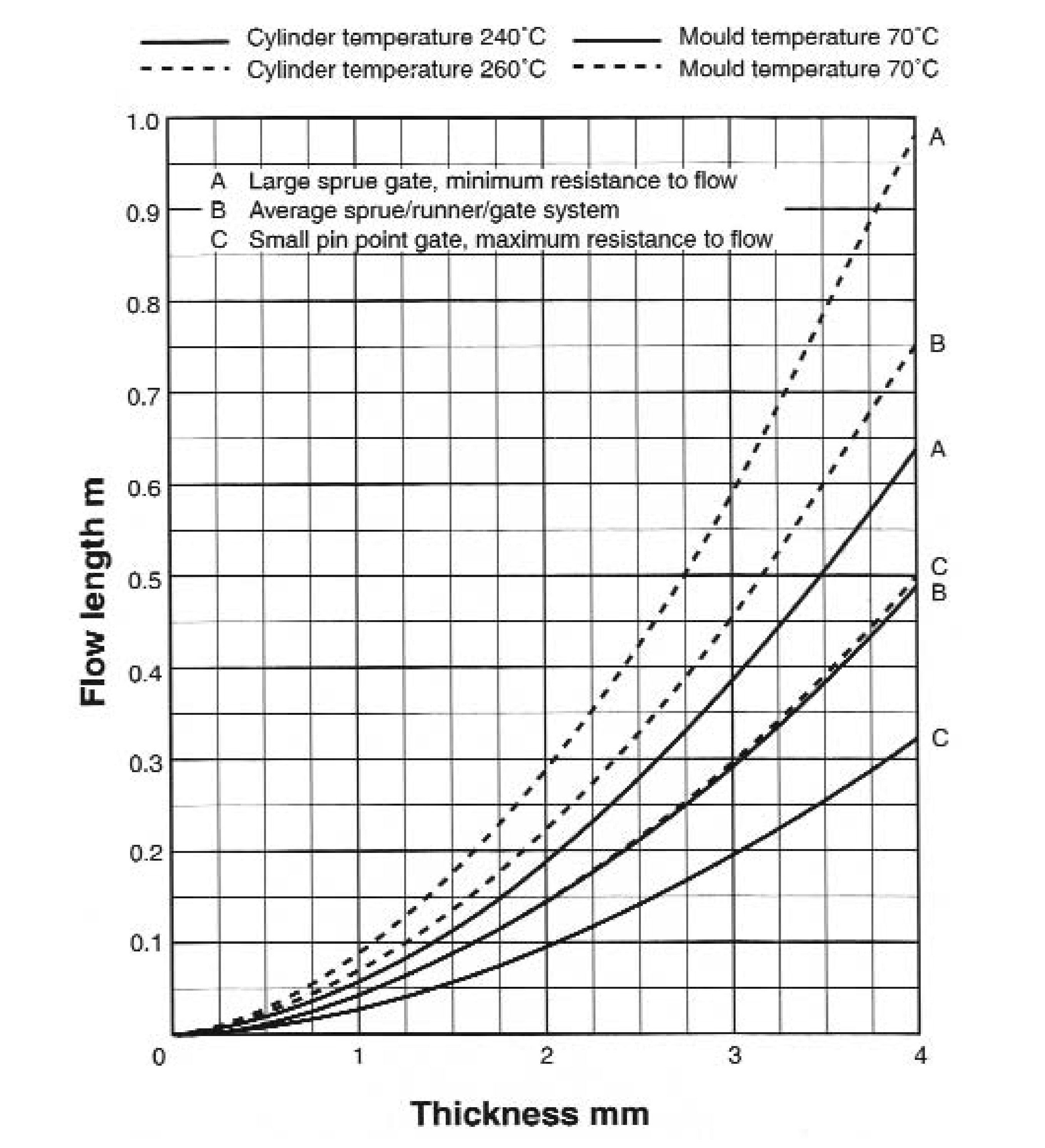


Figure 51 Influence of gate size and melt temperature on the approximate flow ratio for Lucite Diakon CMG302

Cylinder Temperatures

Due to the many factors influencing material or melt temperature it should be noted that melt and cylinder temperatures are unlikely to be identical and in fact may differ by a significant amount.

The approximate range of melt temperature over which Lucite Diakon may be moulded is 200 to 270°C. For average size mouldings the easy flow Lucite Diakon type 6 grades will be in the low to middle range and the higher viscosity Lucite Diakon Type 8 grades in the middle to high range. It is common to optimise temperature settings by applying a small gradient to the cylinder temperature; 5 to 10°C lower at the nozzle and 10 to 20°C lower at the rear or feed zone.

In the absence of experience or correlation between melt and cylinder temperatures then initial cylinder temperature settings of 240°C are recommended.

Mould Temperature

It is essential when moulding Lucite Diakon to have adequate provision for controlling the mould temperature. Both halves of the mould should be cored for circulating water at a controlled temperature. With some mould designs and component shapes it may well be necessary to control the mould halves at different temperatures to achieve an acceptable product. A separate circuit should be used to control the sprue bush temperature.

The recommended mould temperature for the Lucite Diakon type 8 grades is between 60 and 80°C depending upon section thickness and flow path, and for Lucite Diakon type 6 grades 55-70°C.

Machine Start-Up

Injection moulding machines should not be allowed to stand idle for long times while at moulding temperatures, since this allows heat to conduct backwards along the screw and could cause material to melt on to the feed section of the screw and create an obstruction. Where a delay is involved, rear temperature should be temporarily reduced. Controlled water should be circulated around the feed pocket during the heating up period to prevent this section from becoming too hot and causing sticking of prematurely melted material. When in production the feed throat should be maintained between 40 and 60°C.

If any mould setting is required on the injection unit this should be done once the cylinder has attained the moulding temperature. The machine and mould should never be ‘set’ when cold, otherwise the expansion of the injection unit when it reaches moulding temperatures could cause serious damage.

Before commencing to mould, the machine should be purged briefly to ensure that the material in the barrel is clean and at the right temperature.

Screw Back Pressure

When the screw unit is plasticising, a regulated forward hydraulic pressure is applied to the screw in partial opposition to the back pressure generated by the plasticised melt. The regulated pressure is known as the screw back pressure or screw reaction pressure. If this back pressure is greater than the pressure generated by the melt in front of the screw then no screw retraction will take place. However, by adjustment of the screw back pressure, the screw may be made to refill under controlled conditions and produce a uniform melt.

Some back pressure is desirable to help expel air from between the polymer particles or granules and so prevent air from being included in the melt. Otherwise this may lead to burning of the material in the cylinder and may show as splash marks or bubbles (generally with white inclusions) in the moulding, or in the extreme case as black streaks. Screw back pressure is also useful with blends or dry coloured material to aid mixing, particularly where lightly tinted materials are being processed. An increase in screw back pressure causes more work to be done on the material and so enhances mixing. However, excessive use of back pressure can lead to overheating of the material die to frictional heat, which will show as splash marks and could eventually lead to screw slip (see below) due to overheating of material on the rear section of the screw.

Screw Speed

Because acrylic moulding materials have relatively high melt viscosities, attention must be paid to the screw speed to avoid excessive frictional heating and degradation. The screw speeds to be used vary according to the size of machine (ie screw diameter) and type of article being moulded, but in general they should be kept as low as possible consistent with an acceptable cycle time. For shot weights up to 250g screw speeds of 80-100 rpm are used satisfactorily; for machines with large diameter screws it is necessary to keep screw speeds low in the range of 30-40 rpm.

Where temperature controllers indicate a marked tendency to override the preferred set temperature due to frictional heating, then adjustment of screw speed and back pressure should be considered. If full correction by this means is not possible, but the developed temperature can be accepted, then the temperature controller should be re-set to control the temperature at a higher level.

Screw Slip

This term is applied when the screw turns but does not refill. It is generally caused by molten or semi-molten material, in or close to the feed section, sticking to the screw flights and so impeding the entry of fresh material into the cylinder. It can also be caused by too high a screw back pressure.

Screw slip can occasionally occur during start-up. This arises because the machine has been allowed to stand at moulding temperature for too long a time. Under these conditions, heat from the cylinder conducts along the screw raising the temperature of the rear section of the screw which then causes premature melting of material in the feed flights. This is especially so if the screw flights are full of material.

To overcome screw slip, ie remove the blockage caused by molten or semi-molten material, the temperatures of the rear zone and feed pocket should be lowered, insuring cooling water is circulating around the feed throat and the machine purged with rework material. In extreme instances the rework material may have to be force-fed on to the screw. Purging should be continued until the rear temperature stabilises and the screw refills consistently.

Where an extended delay is likely to occur it is a wise precaution to increase cooling to the feed throat and reduce the rear zone cylinder temperature to about 150°C.

Injection Speed

There are contradicting requirements on the rate of filling the mould with acrylic materials. Fast injection speed decreases cycle time, prevents premature freezing of the melt before the mould is full and improves the strength of weld lines. However, with fast injection speed there is a strong possibility of frictional heat and splash marking, especially with small gates, flow lines may be more obvious and there is a higher risk of flashing the mould. Programmed injection allows a balanced rate of fill to be achieved. Fast to medium for the majority of the shot and medium to slow for the balance.

Shrinkage of Mouldings

Shrinkage of mouldings is caused by the reduction in volume which the material undergoes when it changes from the molten to the solid state in the mould and continues to cool to room temperature. The shrinkage expressed as a fraction, or as a percentage, is based on the difference between the dimensions of the cold moulding and of the cold mould. The extent of shrinkage of Lucite Diakon, like that of other thermoplastics, is dependent on the component design, gate design, moulding conditions and the manner in which the melt flows to conform to the shape of the tool.

It is almost impossible to predict accurately the exact amount of shrinkage which will take place on a given article but approximate shrinkage figures which may be used as a guide can be obtained by measurements made on specific test pieces. If accurate dimensions are required on the finished components, it is necessary first to carry out trials under controlled moulding conditions and then to make final adjustments to the mould dimensions.

When doing this it is essential to measure the component sometime after moulding to ensure that full contraction has occurred. The moulding must be kept dry during this time and it is important to measure all critical dimensions both in line with, and across, the flow path of the material, since shrinkage can vary with the direction of melt flow. Shrinkage can be adjusted to some extent by the moulding conditions, but it must be emphasised that the amount of shrinkage which may be controlled in this way is limited and is not always sufficient to compensate entirely for a mould which has been made grossly under or over size. This sort of practice may also lead to the danger of excessive residual stress in a moulding.

On average the shrinkage of Lucite Diakon is in the order of 0.3 to 0.7%, the higher shrinkage applying to a thicker moulding.

From experiments with various components, the following conclusions can be drawn:

Shrinkage is inversely proportional to injection pressure;
Shrinkage is directly proportional to mould temperature;
Shrinkage is directly proportional to melt temperature.

It is worth noting that the flow pattern to the component will tend to determine which is the main factor in controlling the shrinkage of the moulding. For example, the shrinkage of long thin mouldings exhibiting linear flow paths will be dependent more on changes in injection pressure and speed than on other variables, while shrinkage of moulding exhibiting radial flow paths will be more dependent on changes in melt and/or mould temperature.

DISTORTION

Acrylic materials are amorphous and therefore significantly less prone to distortion than crystalline materials. However, distortion or warpage of mouldings can occur and is the result of differential cooling rates; the consequence of incorrect moulding conditions, Figure 52, or component design, Figure 53.

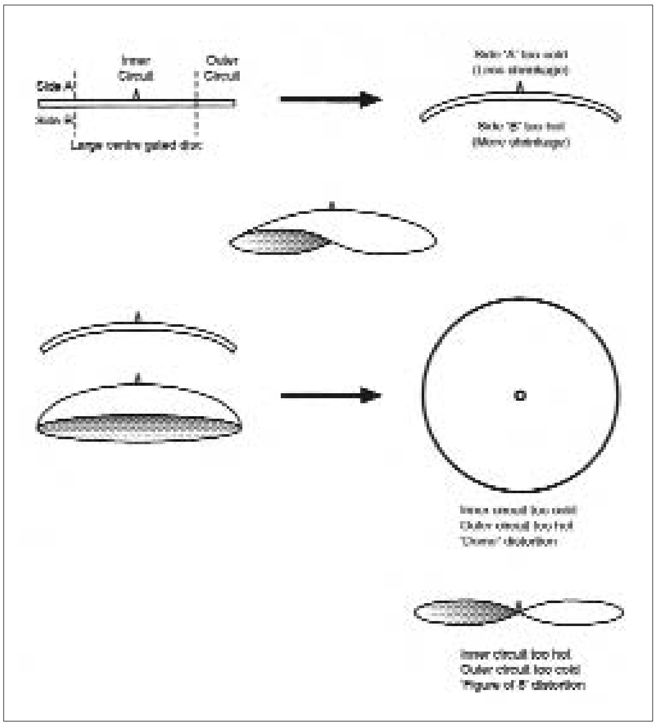


Figure 52 Influence of mould temperature on distortion

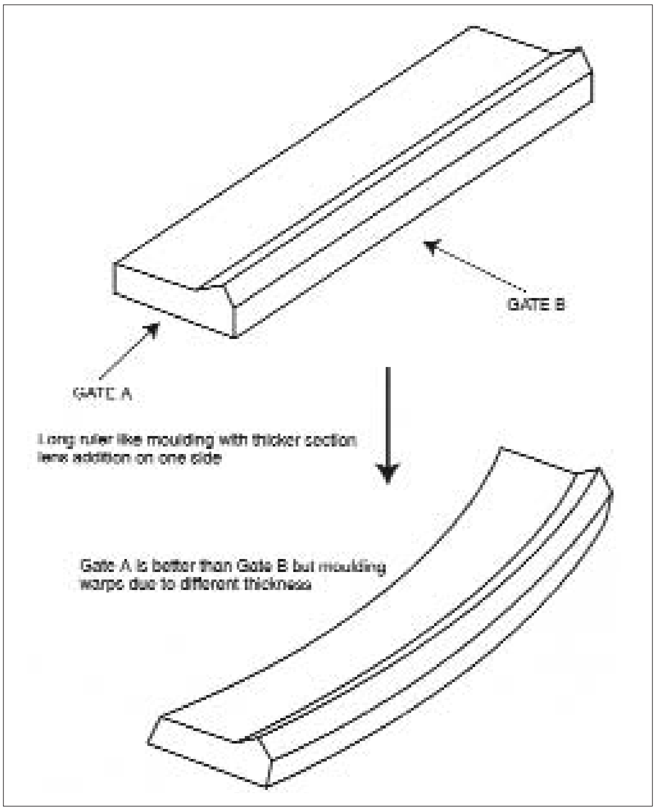


Figure 53 Influence of Component Design on Distortion

Strain in Mouldings

Two types of strain can occur in injection mouldings and these are of consequence in relation to the subsequent service behaviour of the moulded component. These strains arise from:

Molecular orientation - introduced during the flow of the molten polymer in the mould and frozen in during cooling.

Quenching or cooling stress - resulting from a differential rate of cooling between the surface and the interior of the component.

Refer to the section on Stresses and Molecular Orientation in Lucite Diakon Components on page 72 for information on causes, problems, testing and remedy.

MOULDFLOW SIMULATION

Previous sections have described and considered basic principles on the injection moulding of Lucite Diakon; including equipment, component design, mould design and processing conditions. This information has been gleaned from many years of practical experience in the injection moulding of acrylic materials. However significant effort has been put into the development and use of software packages to simulate various aspects of the injection moulding process. Although the initial component design, method of production and subsequent mould design still has to be done, these mouldflow programmes are significant aids to assessment of component design, tool layout including feed system, processing conditions and possible problem areas.

A series of rheology measurements, coefficients and thermal properties for each specific Lucite Diakon grade is required as data input for the software packages. As an illustration, Table 7 provides the information for Lucite Diakon CMG314V, the standard normal molecular weight type 8 grade.

Obs.	Shear Rate	Exp. Visc.	Temp.	Calc.Visc.	Diff.%	Temp. Shift	Std Temp
1.	30.00	7116.00	210.	7835.76	-9.19	7.25	218.99
2.	60.00	4864.00	210.	4782.23	1.71	12.06	221.05
3.	100.00	3454.00	210.	3290.69	4.96	14.42	221.75
4.	150.00	2526.00	210.	2437.05	3.65	14.14	221.68
5.	300.00	1465.00	210.	1452.03	.89	13.20	221.41
6.	600.00	880.00	210.	862.40	2.04	14.37	221.74
7.	1000.00	594.00	210.	586.79	1.23	14.23	221.70
8.	1500.00	442.00	210.	432.10	2.29	15.15	221.94
9.	3000.00	259.00	210.	255.98	1.18	14.90	221.88
10.	6000.00	147.50	210.	151.59	-2.70	12.99	221.34
11.	30.00	4157.00	230.	4238.71	-1.93	1.44	231.79
12.	60.00	2826.00	230.	2845.69	-.69	1.68	232.51
13.	100.00	2089.00	230.	2046.82	2.06	2.01	233.37
14.	150.00	1565.00	230.	1552.29	.82	2.06	233.47
15.	300.00	952.00	230.	948.24	.40	2.21	233.80
16	600.00	577.00	230.	570.52	1.14	2.43	234.23
17.	1000.00	395.00	230.	390.25	1.22	2.53	234.41
18.	1500.00	289.00	230.	288.14	.30	2.49	234.35
19.	3000.00	169.00	230.	171.15	-1.26	2.42	234.22
20.	6000.00	93.00	230.	101.49	-8.37	1.82	232.89
21.	30.00	1720.00	250.	1679.00	2.44	.29	243.42
22.	60.00	1377.00	250.	1351.59	1.88	.31	243.75
23.	100.00	1076.00	250.	1087.84	-1.09	.31	243.79
24.	150.00	860.00	250.	885.93	-2.93	.31	243.87
25.	300.00	570.00	250.	589.62	-3.33	.34	244.39
26.	600.00	373.00	250.	372.92	.02	.43	245.62
27.	1000.00	264.00	250.	260.65	1.29	.49	246.27
28.	1500.00	196.00	250.	194.61	.71	.50	246.39
29.	3000.00	120.00	250.	116.93	2.63	.57	247.13

Table 7 Mouldflow rheology measurements for Lucite Diakon CMG314V

CARREAU EQUATION

REFERENCE TEMPERATURE = 230.00

Fitted activation energy = 165714.93108

Fitted E/R = 19931.07581

THE CARREAU EQUATION

COEFF. P1 = 10369.

COEFF. P2 = .75453e-01

COEFF. P3 = .75629

Fit Coeff. = .99928

Obs.	Shear Rate	Exp. Visc.	Temp.	1st Order Calc. Visc.	1st Order Diff.%	2nd Order Calc.Visc.	2nd Order Diff.%
1.	30.00	7116.00	210.	7111.64	.06	7373.77	-3.50
2.	60.00	4864.00	210.	4394.17	10.69	4760.55	2.17
3.	100.00	3454.00	210.	3081.64	12.08	3390.75	1.87
4.	150.00	2526.00	210.	2325.25	8.63	2563.96	-1.48
5.	300.00	1465.00	210.	1436.74	1.97	1557.21	-5.92
6.	600.00	880.00	210.	887.74	-.87	921.21	-4.47
7.	1000.00	594.00	210.	622.57	-4.59	615.21	-3.45
8.	1500.00	442.00	210.	469.76	-5.91	442.01	.00
9.	3000.00	259.00	210.	290.26	-10.77	245.99	5.29
10.	6000.00	147.50	210.	179.35	-17.76	133.34	10.62
11.	30.00	4157.00	230.	4331.19	-4.02	3923.97	5.94
12.	60.00	2826.00	230.	2676.18	5.60	2648.24	6.71
13.	100.00	2089.00	230.	1876.81	11.31	1948.91	7.19
14.	150.00	1565.00	230.	1416.15	10.51	1512.44	3.48
15.	300.00	952.00	230.	875.01	8.80	960.23	-.86
16	600.00	577.00	230.	540.66	6.72	593.82	-2.83
17.	1000.00	395.00	230.	379.16	4.18	409.75	-3.60
18.	1500.00	289.00	230.	286.10	1.01	302.13	-4.35
19.	3000.00	169.00	230.	176.78	-4.40	175.77	-3.85
20.	6000.00	93.00	230.	109.23	-14.86	99.60	-6.63
21.	30.00	1720.00	250.	2637.82	-34.79	1968.00	-12.60
22.	60.00	1377.00	250.	1629.87	-15.51	1388.42	-.82
23.	100.00	1076.00	250.	1143.03	-5.86	1055.73	1.92
24.	150.00	860.00	250.	862.47	-.29	840.83	2.28
25.	300.00	570.00	250.	532.91	6.96	558.05	2.14
26.	600.00	373.00	250.	329.28	13.28	360.75	3.40
27.	1000.00	264.00	250.	230.92	14.32	257.20	2.64
28.	1500.00	196.00	250.	174.24	12.49	194.63	.70
29.	3000.00	120.00	250.	107.66	11.46	118.37	1.38

1st Order (Power Law)	Coeff.A =	.13781E+08
	Coeff.B =	-.69459
	Coeff.C =	-.24794E-01
Fit Coeff. = .98896		
2nd Order (Quadratic)	Coeff.A1 =	16.003
	Coeff.A2 =	-1.0980
	Coeff.A3 =	-.98320E-02
	Coeff.A4 =	-.27380E-01
	Coeff.A5 =	.31997E-02
	Coeff.A6 =	-.74073E-04
Fit Coeff. = .99827		
Specific heat capacity	2300 J/kg deg C	
Thermal conductivity	0.2 w/m deg C	
Melt density	1100 kg/mx3	
No flow temperature	160 deg C	
Freeze temperature	120 deg C	

A joint exercise between Lucite International and Plastics Design Solutions Ltd (See Appendix II for full address) was carried out to illustrate how data for the mouldflow simulation process may be used to influence the design optimisation and production of Lucite Diakon mouldings. The simple component design representing an instrument panel lens together with cavity, sprue, runner, gate size and gate position variations is illustrated in Figures 54 and 55. A matrix of mouldflow data obtained from these variations together with changes in material and processing conditions is listed in Table 8.

Table 8 Summary of conditions and results

LUCITE DIAKON® Grade	Cavity Thickness mm	Gate Size mm	Gate Position	Sprue and Runner	Injection Time seconds	Melt Flow Rate cm^3sec^{-1}	Melt Temp. °C	Apparent Bulk Melt Temp. °C	Maximum Pressure Bar	Maximum Shear Stress KPa	Maximum Shear at Gate sec-1	Average Shear Rate at Gate sec-1	Comment
CMG314V	2.2	1 x 5	B	Good	2	65	250	-	1,793	776	47,100	-	infuence
CMG314V	2.5	1 x 5	B	Good	2	73	250	-	1,505	796	51,600	-	of cavity
CMG314V	2.8	1 x 5	B	Good	2	81	250	-	1,321	812	56,800	-	thickness
CMG314V	2.5	1 x 5	A	Good	2	68	250	-	967	750	48,400	-	see fig 56
CMG314V	2.5	1 x 5	B	Good	2	73	250	-	1,505	796	51,600	-	
CMG314V	2.5	1 x 5	C	Good	2	81	250	-	1,808	818	56,700	-	
CMG314V	2.5	1 x 5	B	Good	2	73	250	252	1,505	796	51,600	-	see fig 57
CMG314V	2.5	1 x 5	B	Poor	2	68	250	283	2,135	746	46,200	-	
CMG314V	2.5	1 x 5	B	Good	0.5	146	250	-	1,795	1,010	200,900	-	see fig 58
CMG314V	2.5	1 x 5	B	Good	1.7	86	250	-	1,532	818	60,300	-	
CMG314V	2.5	1 x 5	B	Good	3	36	250	-	1,725	742	35,800	-	
CMG314V	2.5	1 x 5	B	Good	2	73	230	-	2,050	994	-	-	see fig 60
CMG314V	2.5	1 x 5	B	Good	2	73	250	-	1,505	796	-	-	
CMG314V	2.5	1 x 5	B	Good	2	73	270	-	1,067	661	-	-	
CMG314V	2.5	1 x 2	B	Good	2	73	250	-	1,604	1,010	204,200	169,400	see fig 59
CMG314V	2.5	1 x 5	B	Good	2	73	250	-	1,505	796	51,600	44,500	
CMG314V	2.5	2 x 5	B	Good	2	73	250	-	1,423	672	18,100	16,400	
CMH454	2.5	1 x 5	B	Good	2	73	250	-	1,596	816	-	-	infuence
CMG314V	2.5	1 x 5	B	Good	2	73	250	-	1,505	796	-	-	of Lucite Diakon
CLG356	2.5	1 x 5	B	Good	2	73	250	-	502	584	-	-	grade

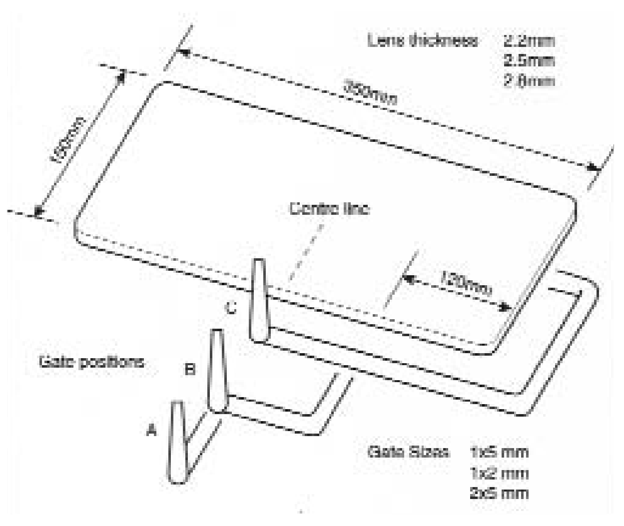


Figure 54 Component and mould cavity layout

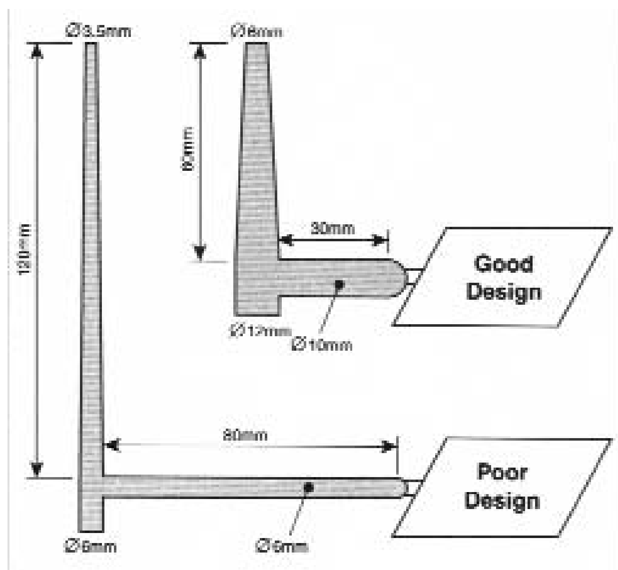
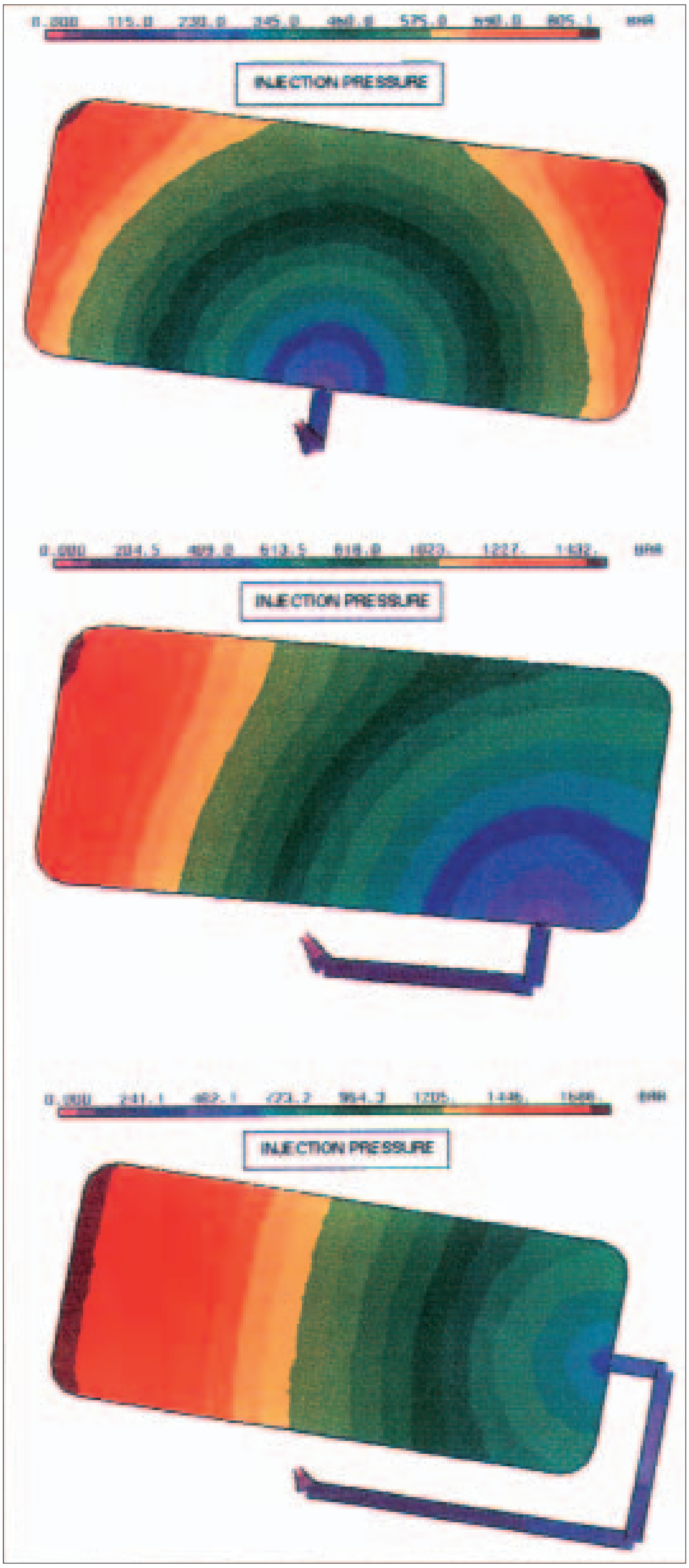


Figure 55 Sprue and runner designs

Figure 56 Influence of gate position on injection pressure, illustrates that cavity layout has a significant effect on the required injection pressure and resultant machine size through locking force requirement. Similar plots are obtained for filling pattern and material shear stress. These may indicate possible distortion, air entrapment or frictional heating problems



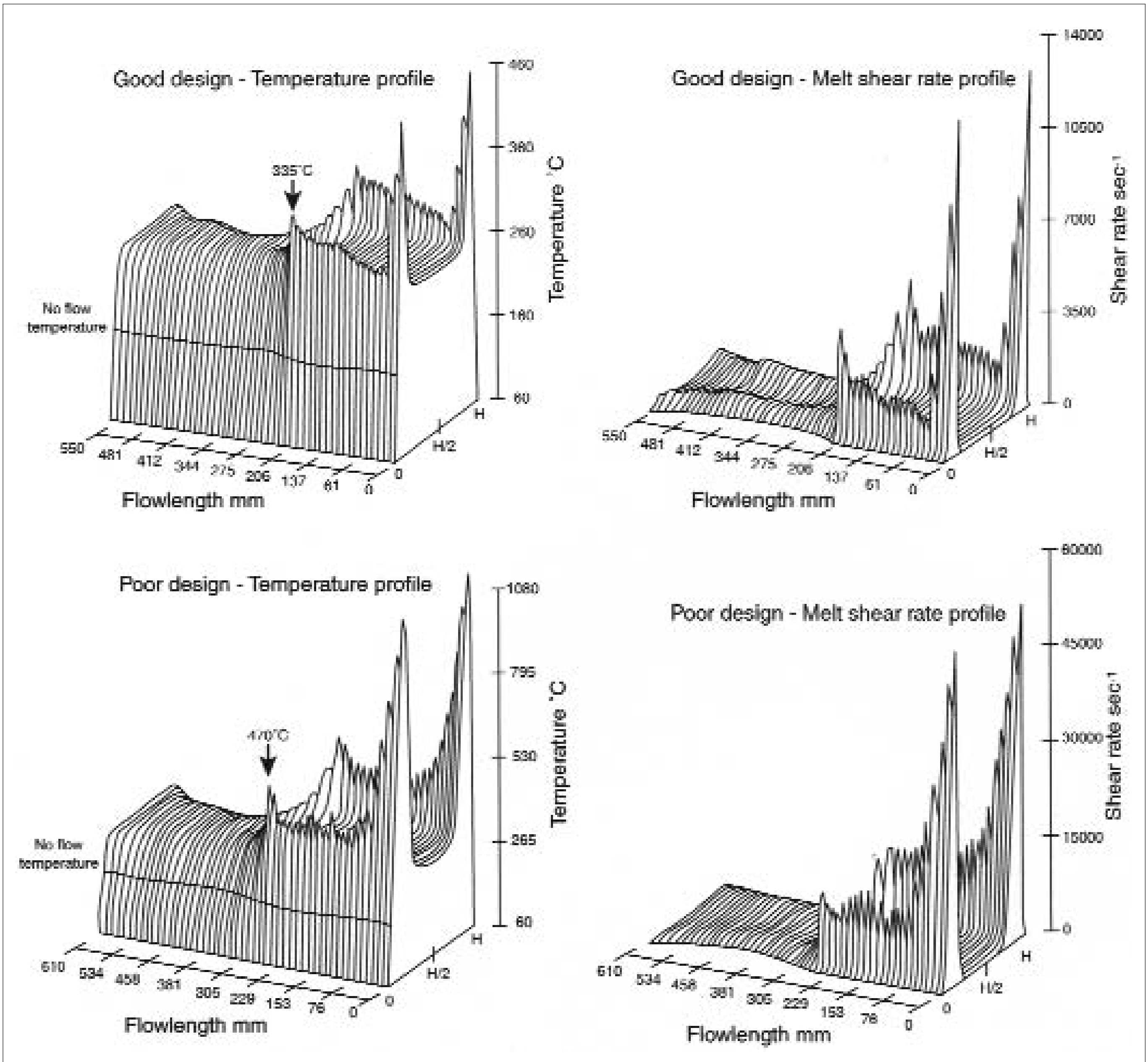


Figure 57 Influence of sprue and runner design on temperature and shear rate profiles. As indicated in other sections on design and processing, the comparatively high melt viscosity and shear sensitivity of acrylic materials may lead to overheating, degradation and monomer splashing. Although the figures produced in a simulation exercise may not be exactly those obtained in practice, the results in Figure 57 strongly underline the difference between good and bad design in the feed system. It also illustrates that it is often the feed system and not the mould cavity that controls and influences the injection moulding process. There are many occasions where mouldflow exercises are limited to the actual component cavity but for the purpose of production efficiency it is recommended that consideration is given to modelling the feed system

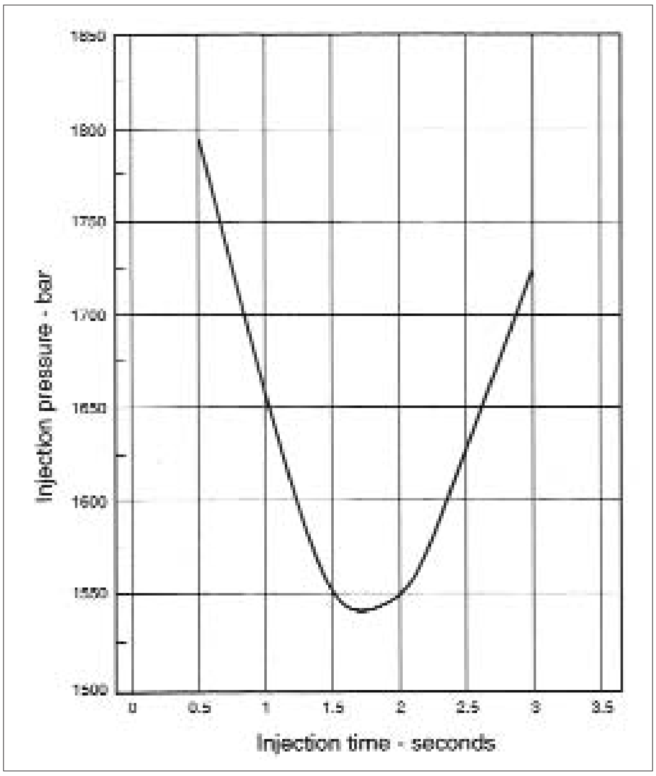


Figure 58 Influence of injection time on injection pressure, shows that there is an optimum fill time to minimise injection pressure although other factors like degree of sinking may influence this

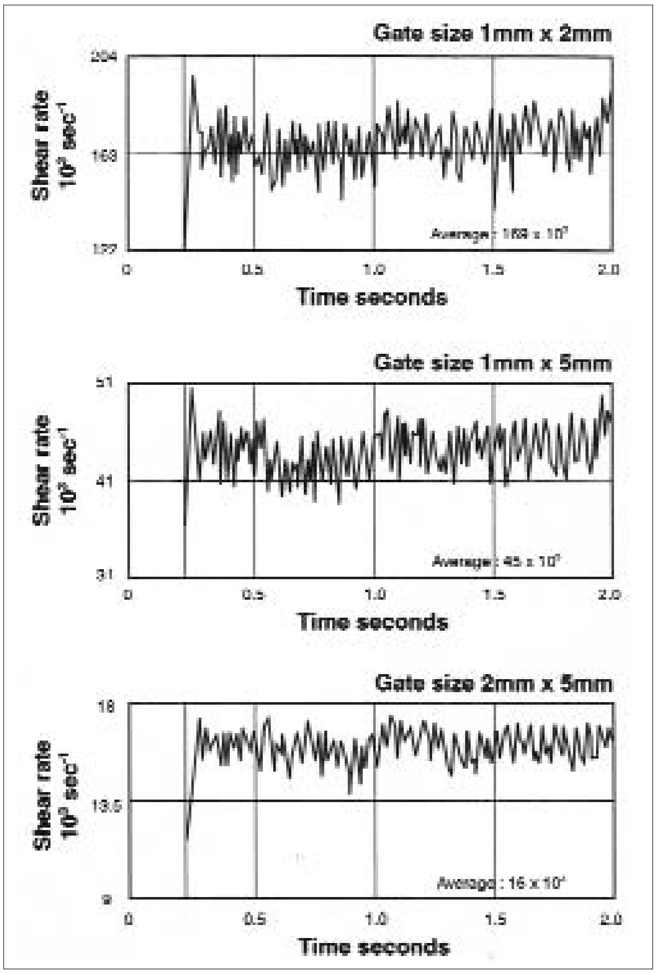


Figure 59 Influence of gate size on shear rate at the gate, aptly illustrates the frictional shear degradation problems that often occurs on the injection moulding of acrylic materials due to the use of small gates

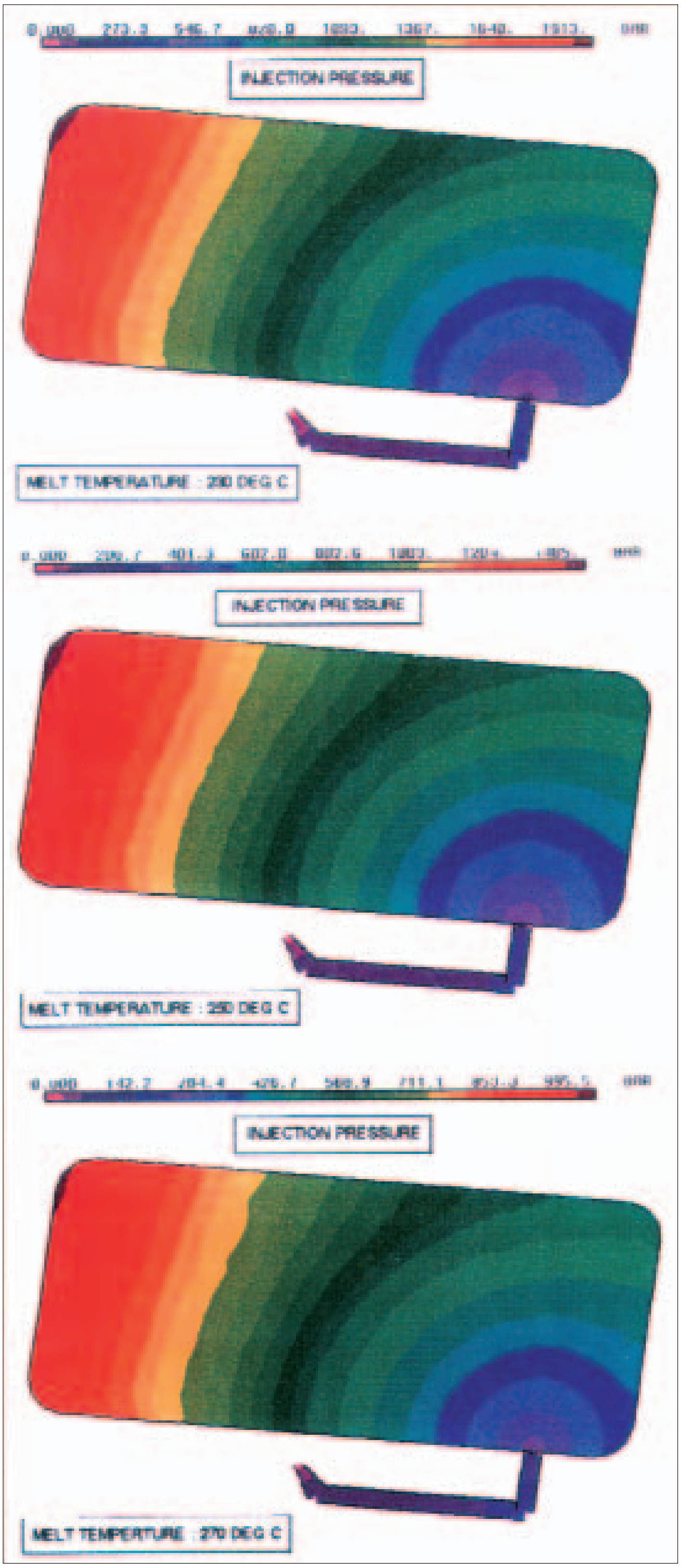


Figure 60 Influence of melt temperature on injection pressure. This figure is included to demonstrate how the mouldflow simulation may be used to influence processing conditions

MOULDING FAULT REMEDIES

The following table lists the main moulding faults likely to be encountered, their causes and the procedures to be followed in order to correct the faults.

Moulding Fault	Remedy
Splash or mica marks, surface streaks These are caused by volatiles (moisture, monomer) in the melt. A major source of degradation and splash marks is excess frictional heat.	**A Too Hot** 1 Reduce cylinder temperature 2 Reduce hot runner temperature 3 Reduce screw speed 4 Reduce back pressure 5 Reduce injection speed 6 Increase size of sprue/runner/gate **B Too cold** 1 Raise cylinder temperature 2 Raise nozzle temperature **C Moisture** 1 Dry the material (70 to 80°C for 6 to 12 hours)
Burning or entrapment of air in cylinder This usually appears as splash marks and small bubbles with white inclusions. In its severest form as black streaks.	1 Increase back pressure 2 Decrease screw speed
Burn marks on moulding Usually appear on extremities of the moulding and are caused by insufficient venting of the cavities.	1 Reduce injection speed 2 Reduce injection pressure 3 Reduce mould locking pressure 4 Reduce cylinder temperature 5 Improve venting of cavity
Matt patches on moulding surface These generally occur in the same position on each moulding, usually close to the gate area. Often caused by a cold slug from the nozzle.	1 Check nozzle seating for dribble 2 If using vacuum suck-back check operation 3 If nozzle has mechanical shut-off check operation 4 Increase nozzle temperature 5 Incorporate cold slug-well opposite sprue or enlarge existing one 6 Polish runner and gate
'Orange peel' and smudge marks Surface imperfections resembling orange peel that occur in the gate area.	1 Reduce cylinder temperature 2 Reduce mould temperature 3 Reduce injection time 4 Increase injection speed 5 Reduce injection pressure 6 Examine gate area for roughness, and polish if necessary
Voids and sink marks These are usually due to insufficient pressure to counterbalance material shrinkage in thick sections or in sections furthermost from the gate.	1 Check feed setting 2 Increase injection pressure 3 Increase injection time 4 Increase injection speed 5 Reduce mould temperature 6 Reduce cylinder temperature 7 Enlarge gate, sprue or runner to reduce pressure loss

Moulding Fault	Remedy
Short shot (incomplete filling of mould) or rippled surface This usually occurs in an area furthest from the gate. It is usually accompanied by a rippled surface in the area surrounding the short, shot.	1 Check feed setting. Make sure sufficient material available 2 Increase injection speed 3 Increase injection pressure 4 Increase injection forward time 5 Increase mould temperature 6 Increase cylinder temperature 7 Enlarge gate, sprue or runners to reduce pressure loss.
Warping Caused by uneven shrinkage in the moulding. Occurs particularly on flatmouldings or mouldings with long edges. Also found on mouldings of uneven section.	1 Increase cooling time 2 Use even (both sides) mould temperatures for flat mouldings 3 Use differential mould temperature control over mould surfaces, or between mould halves where opposite surface areas differ. 4 Adjust injection speed. 5 Reduce cylinder temperatures. 6 Use clamping jig in which to cool mouldings.
Weld lines, flow lines These are caused by the melt separating and rejoining in the mould. They usually occur around inserts or as tails from raised characters of sections	1 Increase injection pressure 2 Reduce injection speed (Occasionally, to eliminate weld lines, it may be necessary to increase injection speed) 3 Increase mould temperature 4 Increase cylinder temperature 5 Change location of gate to alter flow pattern 6 Radius corners to improve flow in mould
Jetting or flow line Usually occur in gate area	1 Reduce injection speed 2 Reduce cylinder temperature 3 Use tab gate
Crazing This occurs as minute surface cracks on the moulding, usually in line of flow	1 Clean mould surface 2 Increase injection speed 3 Increase mould temperature
Delamination This usually occurs in the gate area or as blisters on the moulding surface	1 Check for contamination
Cracking or breaking of the part on ejection	1 Decrease injection pressure 2 Decrease injection time 3 Mould opening and ejection speed 4 Increase mould temperature 5 Increase draft angle 6 Eliminate sharp corners and undercuts.