Qenos

05

INJECTION
MOULDING
—

TECHNICAL GUIDE

Alkathene®   Alkatuff®   Alkatane®

Front Cover

Qenos produces injection moulded products for applications including caps, pails, crates, sealant cartridges, mobile garbage bins, produce bins, housewares and lids. A full range of *Alkatane* HDPE, *Alkathene* LDPE and *Alkatuff* LLDPE grades are available across the Melt Index and density spectrum. In addition, Qenos distributes a number of speciality polymers suitable for injection moulding.

Qenos, the Qenos brandmark, *Alkathene*, *Alkatuff*, *Alkamax*, *Alkadyne* and *Alkatane* are trade marks of Qenos Pty Ltd.

INJECTION
MOULDING

5

5 INJECTION MOULDING

INTRODUCTION

The purpose of this document is to provide an introduction to the processing of polyethylene by injection moulding. The effects of Melt Flow Index (MFI) and density on moulding characteristics and on the properties of the finished moulding are discussed, in the light of which, recommendations are made as to the desirable values of these two factors for stressed and unstressed applications.

Mould design is considered with special reference to questions of shrinkage and distortion and examples are given to illustrate these points. The moulding process itself is discussed in some detail, guidance being given on all the operations which have to be carried out. Moulding faults, causes and remedies are also summarised.

5 INJECTION MOULDING

INTRODUCTION

Injection moulding is one of the most widely used processes for converting thermoplastic raw materials into finished products. Fundamentally, a solid polymer is plasticated into a molten mass via thermal and frictional heating and once a suitable volume of melt has been produced, the polymer is injected into the mould to form the finished part (see Figures 1 and 2).
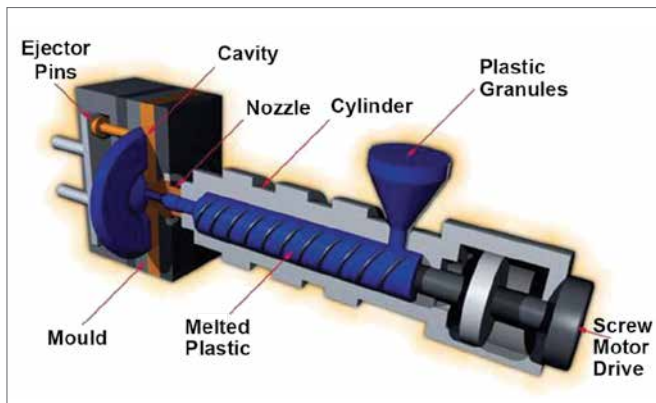
Figure 1: Schematic Representation of an Injection Moulding Machine

Figure 2: Finished Moulded Part including Sprue Injection Point

Injection Moulding is fundamentally simple, easy to operate and is capable of producing a very wide variety of industrial and domestic articles. Of all thermoplastics, polyethylene is one of the easiest to injection mould. The resin flows easily into difficult cavities, its viscosity changes smoothly as the melt temperature increases and it can be processed over a wide temperature range without decomposition.

However, this very ease of processing often leads to the use of moulding conditions which are not the most suitable for producing the finished part. Also, because almost all of the many different types of polyethylene can be moulded on standard equipment, the polyethylene type that is most suitable for a particular application is not always chosen.

EFFECT OF TYPE OF POLYETHYLENE ON PROCESSING AND PROPERTIES OF MOULDINGS

To obtain polyethylene mouldings which will withstand long and arduous service two important questions must be answered:

a. Which type of polyethylene should be used?

b. What are the correct moulding conditions?

To do this it is necessary to know how the different types of polyethylene used for injection moulding differ from each other: first, in the way in which they are processed and second, in the physical properties of the moulded article.

Classification of Polyethylenes

The most important variables which characterise a polyethylene are its Melt Flow Index (MFI) and density.

Melt Flow Index (MFI)

MFI is a measure of melt viscosity at low shear rates and is defined as the weight in grams of polyethylene extruded in 10 minutes from a special plastometer under a given load at 190°C. Thus, a low MFI corresponds to a high melt viscosity. Figure 3 shows how the MFI is related to the number average molecular weight of the polymer.

Figure 3: Relation between MFI (g/10 min) and Number Average Molecular Weight

DENSITY

Density is related to the crystallinity of the polyethylene and is measured in g/cm^3.

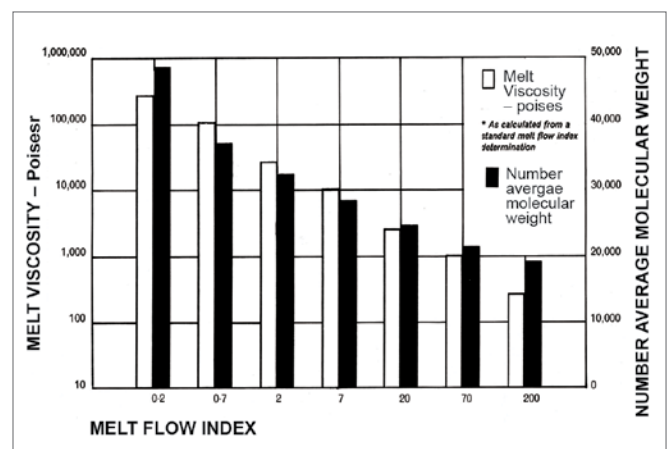Because polyethylene molecules are long and contain branches, complete crystallisation cannot take place when polyethylene is cooled from the molten state, and amorphous regions occur between the crystallites. The smaller the number of branches, the more crystalline the polyethylene will be and the higher its density.

Although MFI and density are the most important variables which characterise a polyethylene, it must be emphasised that all polyethylenes with the same MFI and density are not necessarily identical. Each polyethylene producer has specific manufacturing processes and by varying reactor conditions it is possible, while maintaining a constant MFI and density, to alter various features of the polymer such as the molecular weight distribution and the degree of long and short chain branching that cause changes in the processing behaviour and the physical properties of the polymer.

Effect of MFI and Density on Moulding Characteristics

The injection moulding process is shown diagrammatically in Figures 4 and 5. For any given machine and mould, the MFI and density of the polyethylene will considerably affect the injection dwell and cooling times in the cycle. The injection time is not significantly affected and the mould opening, extraction, and mould closing times are not affected by the MFI or density of the polymer.
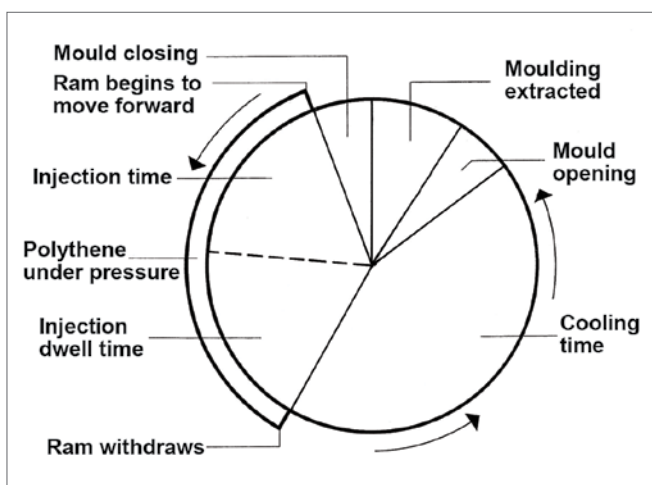
Figure 4: **Injection Moulding Cycle**
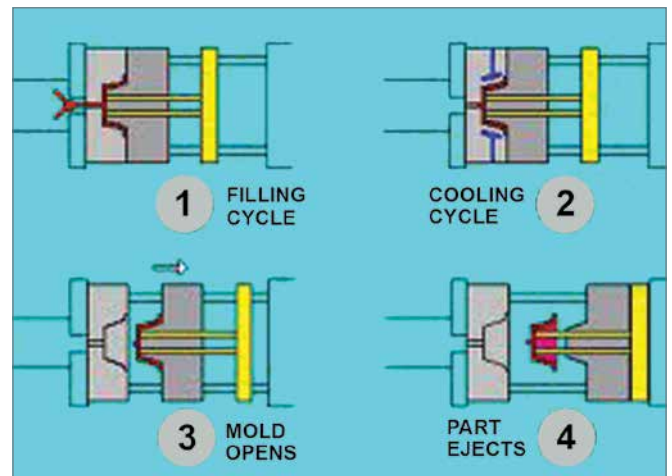
Figure 5: **Pictorial Representation of the Injection Moulding Cycle**

As far as the polyethylene is concerned the output of any injection moulding machine depends predominantly on two factors:

- The time taken for the polyethylene to reach moulding temperature
- The time taken for the polymer to be cooled sufficiently in order for the moulding to be removed.

A convenient method of assessing the effect of different types of polyethylene on output rate is to plot the number of mouldings which can be made in one hour against the cylinder temperature used. Although the design of the mould and the type of machine affect output greatly, for any given mould on a particular machine an output curve can be obtained by finding for each cylinder temperature the fastest possible cycle which gives mouldings acceptable in all respects except for that of surface gloss, i.e. the minimum injection dwell time, pressure, and cooling time have been used. A typical curve for a plunger machine is shown in Figure 6. It will be noticed that, at first, as the temperature increases the output also increases. The reason for this is that at low temperatures a long cycle is necessary to melt the granules thoroughly, but as the temperature increases, the melting time becomes shorter and therefore the cycle is also shortened. A point is soon reached, however, when the time taken to melt the granules is no longer the limiting factor. The greater parameter of importance is then the time taken for the mouldings to cool to a temperature at which they can be extracted easily from the mould. Beyond this point, as the melt temperature increases the cycle time has to be extended and the output consequently falls.

5 INJECTION MOULDING

Figure 6: Variation in Output Rate of Mouldings with Cylinder Temperature

Figure 7 shows the effect of density on output rate for polyethylenes of the same MFI. It indicates that the higher the density, the higher the output rate on the cooling side of the curve at any given cylinder temperature. The reason for this is that mouldings of higher density can be extracted from the mould at higher temperatures because they are more rigid at these temperatures than are mouldings of lower density. The higher density materials, however, require higher cylinder temperatures to produce adequate melting of the granules, particularly if the amount of material being handled is near the plasticising capacity of the machine, and the use of such temperatures may slow down the output rate.
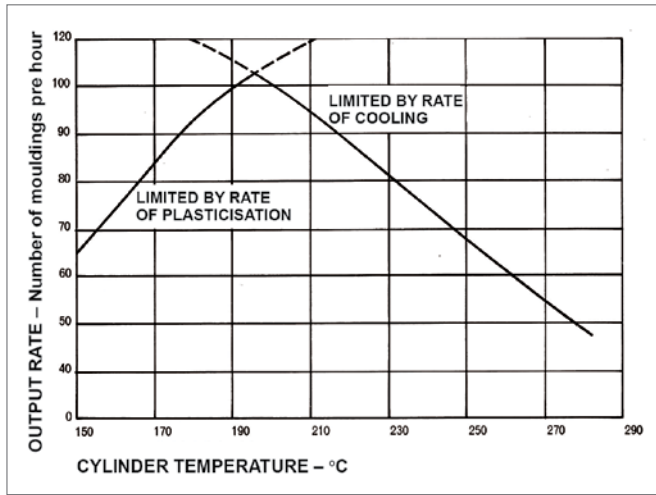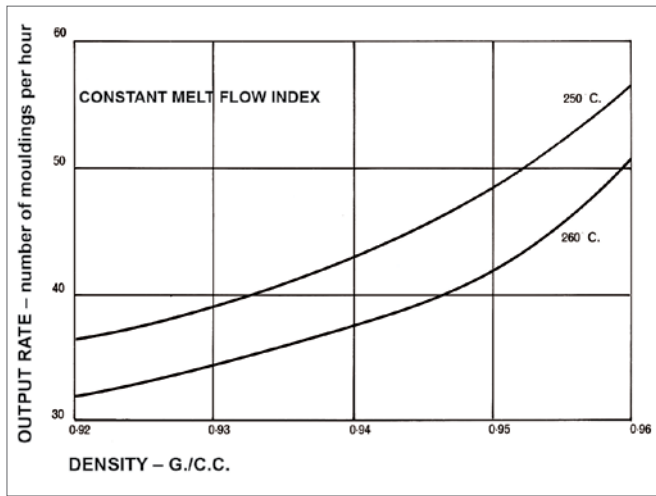

Figure 7: Effect of Density on Output Rate for Polymers of the Same MFI

To use injection moulding machines most efficiently, the cylinder temperature should be chosen so that the output is at its peak. There are, however, two factors which frequently prevent this being done, namely, the necessity to fill the mould, and the desire to obtain mouldings with a good surface finish. These factors are discussed below.

MOULD FILLING

In practice, there are some moulds for which it is not possible to draw an output curve over the whole range of cylinder temperatures because the mould cannot be filled at the lower temperatures. Therefore, the moulding temperature which has to be used is the lowest temperature at which the mould can be filled, and this may restrict the output. In order to attain as close to the maximum theoretical output, good mould filling properties are obviously desirable in a polyethylene.

The spiral flow test was devised to assess the mould filling properties of materials. It involves the measurement of the length of spiral obtained when moulding under standard conditions using the special mould shown in Figure 8. In order to compare different types of polyethylene the cylinder temperature, mould temperature, cycle time, injection speed and pressure are all held constant, and under these conditions the length of spiral obtained gives a good comparative evaluation of the mould filling properties of the polyethylenes being used.
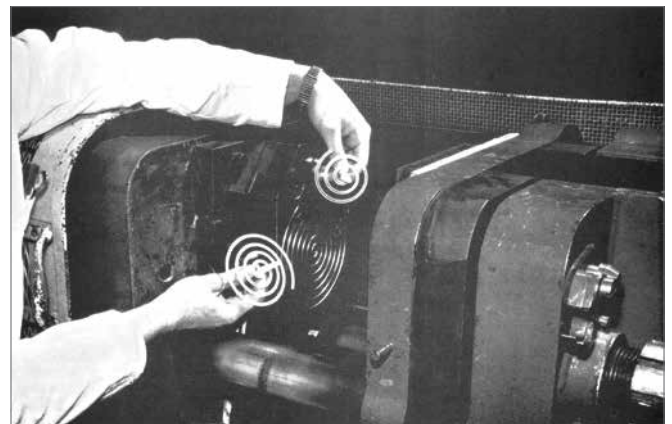
Figure 8: Spiral Flow Mould

Figure 9 shows that the main factor which influences ease of mould filling is MFI. Although density undoubtedly has an effect on the spiral flow length, for polymers with constant MFI this effect is relatively small.
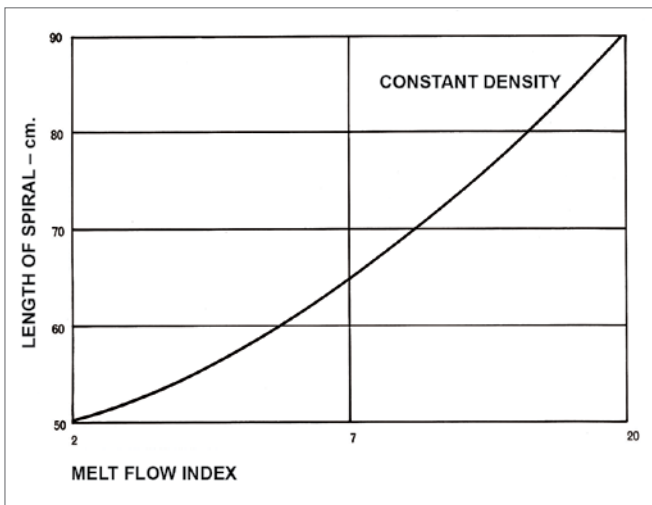
Figure 9: Effect of MFI on the Mould Filling Properties of Polyethylenes of Constant Density

A feature of the spiral flow test is that it can be applied to all injection moulding materials. Figure 10 shows a chart on which the spiral flow length has been plotted against a series of cylinder temperatures for a range of polymers. For most materials the temperatures used range from the lowest at which a readable flow length can be obtained to the highest that can be used without degrading the material. However for polyethylenes of high MFI, with the particular equipment used, the upper temperature was set by the first observance of "flashing" (thin films of excess polymer) on the moulded part.
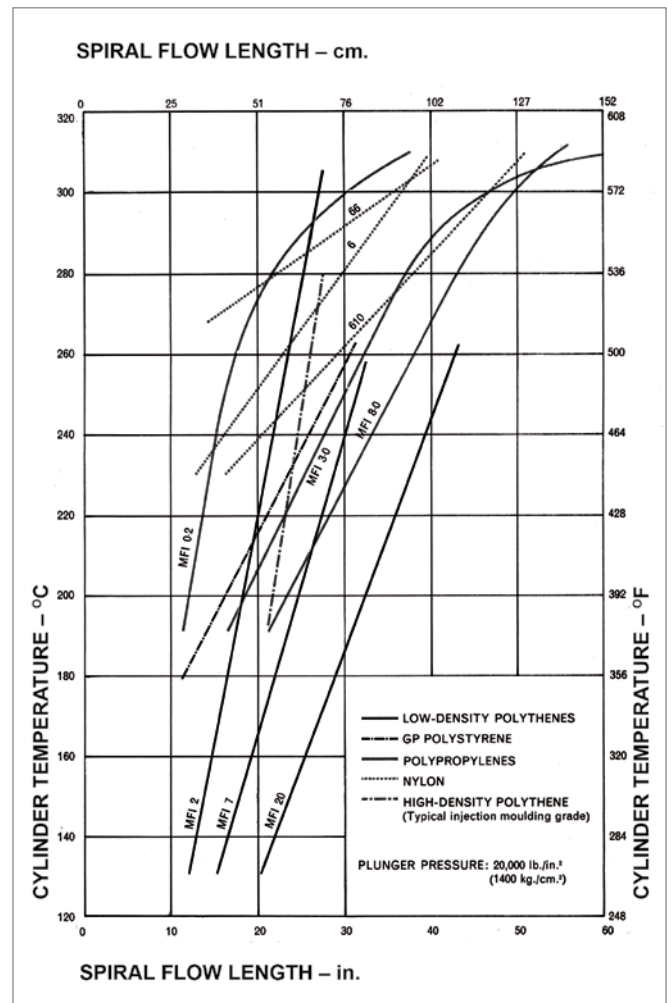
Figure 10: Spiral Flow Curves for some Typical Thermoplastics

Surface Finish

The second factor which may prevent moulding being carried out at the peak of the output curve is the requirement to obtain a good surface finish on the moulded article. It can be seen from Figure 11 that the gloss of a polyethylene moulding improves with increasing cylinder temperature and that mouldings produced at the lower temperatures have 'chevron' marks or rings on the surface (see Figure 12). When mouldings with an even, glossy surface are required it may be necessary to mould at a cylinder temperature which is higher than that which corresponds to the fastest output rate.

5 INJECTION MOULDING
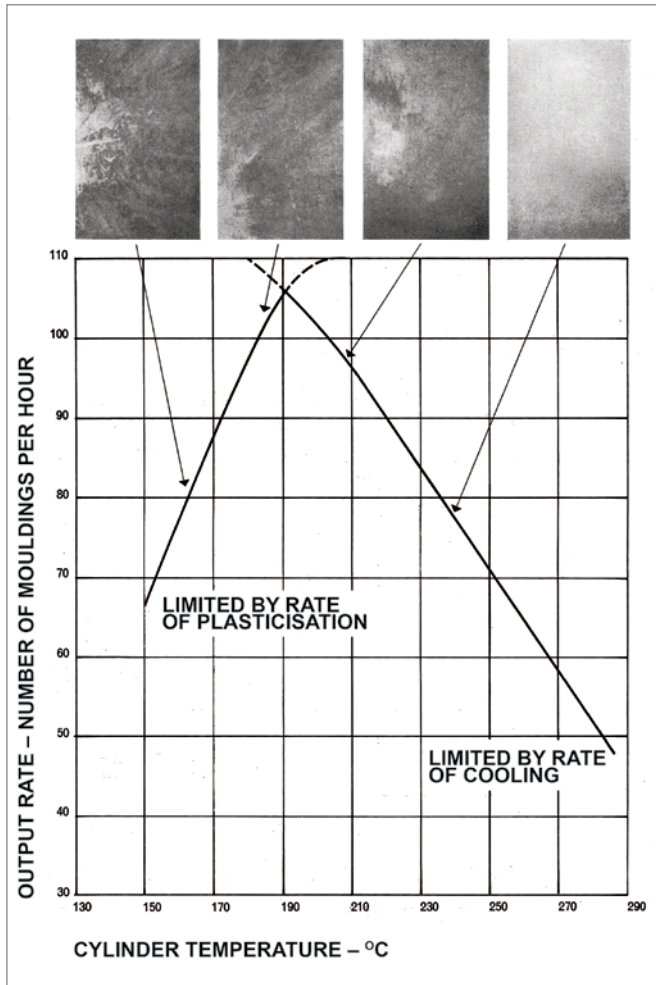
Figure 11: Variations of Surface Gloss of Mouldings with Cylinder Temperature

Gloss is assessed both visually and by measuring the light reflected from the surface of mouldings made under standard conditions. By the latter method, gloss/temperature curves can be plotted as shown in Figure 13. This not only shows the effect of cylinder temperature on gloss, but also the very marked effect of MFI. With a higher MFI, high-gloss mouldings can be produced at a lower cylinder temperature which allows for a faster output (see Figure 13).

Figure 12: Photo Illustrating 'Chevron' Rings on an Injection Moulded Surface
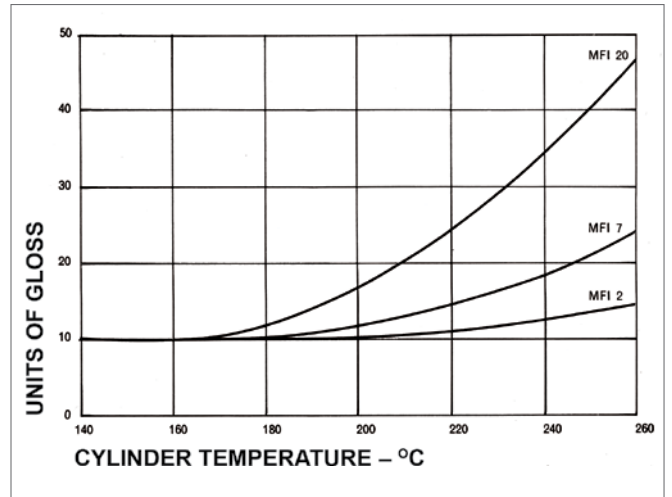
Figure 13: Effect of MFI and Temperature on Gloss

Summary

It can be concluded that a high MFI is the characteristic mainly responsible for ease of moulding and high output rates. The higher the MFI, the lower the cylinder temperature which can be used to obtain adequate mould filling and acceptable surface finish, and consequently, in most cases, the higher the output will be. For resins with a constant MFI, the degree to which an increase in density leads to higher or lower outputs will depend mainly on the size of the moulding in relation to the size of the machine. For adequate melting of the granules, higher density polyethylenes require higher cylinder temperatures than do the lower density polyethylenes, and melting is more likely to be a limiting factor.

Thus, as far as processing is concerned, the type of polyethylene chosen should have as high an MFI as possible. However, the choice of both MFI and density must also take into account the physical properties required in the finished moulding, and this subject is discussed in the next section.

EFFECT OF MFI AND DENSITY ON THE PROPERTIES OF POLYETHYLENE MOULDINGS

The physical properties of polyethylene which are of particular importance in injection moulded articles are:

- Stiffness
- Impact properties
- Resistance to environmental stress cracking
- Resistance to mechanical stress cracking

Stiffness

The main factor determining the stiffness of a moulding is the density of the polyethylene. Figure 14 shows how the stiffness (as measured by the 100 sec tensile modulus) increases rapidly with increasing density. In the lower density range a change in density of as little as 0.007 g/cm^3 will double the stiffness. Figure 14 also shows the effect of temperature on stiffness.

MFI has virtually no effect on stiffness.
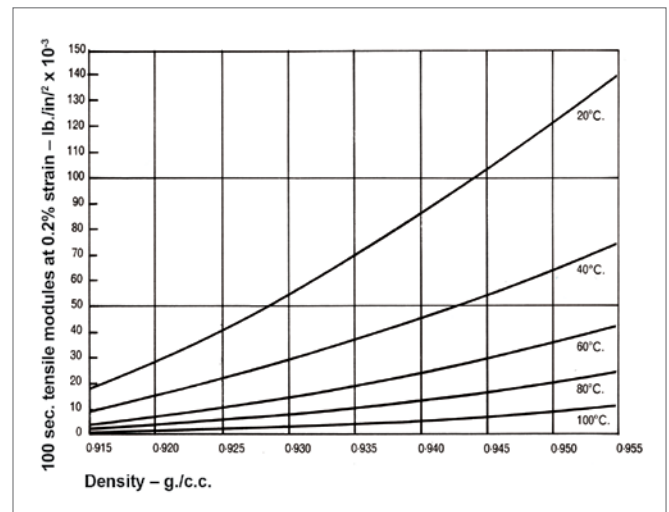
Figure 14: Variation of Stiffness and Density with Temperature

Impact Properties

One of the outstanding properties of low density polyethylene is its toughness; when subjected to impact it will stretch and cold-draw before it breaks, rather than fail in a glass-like manner. On the other hand, medium and high density polyethylenes can fail in a way that is unknown in low density polyethylenes. This type of failure is known as brittle failure. It is quite different from the tough failure of low density materials and is particularly noticeable in mouldings which have sharp notches or scratches on the surface. The usual impact tests for plastic materials are difficult to apply to both brittle and tough types of polyethylene and therefore a special test had to be devised. For this an impact machine is used (see Figure 15) in which small specimens (1 x 1 x 0.16 cm) are notched to a depth of 0.020 cm and subjected to a blow from a pendulum. The energy lost by the pendulum in striking the specimens is termed the impact energy, although much of this energy is expended in bending the specimen as the pendulum swings past it. Polyethylene specimens are rarely broken by the first blow, and therefore after a short rest period they are given a second blow. The energy absorbed by this second blow, expressed as a percentage of the energy absorbed by the first blow, is termed the fracture resistance. This quantity is found to be a useful measure of the amount of damage caused by the first blow.

Figure 15: Impact Machine Showing Sample Holder and Process of Use

Impact energy and fracture resistance depend on both MFI and density, as may be seen from Figure 16. For some polyethylenes the impact energy may increase at first with increasing density and then decrease. This initial increase in impact energy is due to the contribution from the energy used in bending a specimen of increased stiffness. Ultimately, however, the increase in density trends towards brittleness, which becomes the dominant factor and results in the measured impact energy falling to very low levels. It can be seen quite clearly that in order to avoid brittleness the higher density polyethylenes must have a low MFI. Consequently, if toughness is required in the higher density polyethylenes, poorer processability, poorer mould filling and, in general, higher processing temperatures will be required. It can also be seen that with polyethylenes of lower density, a much wider choice of MFI is possible without sacrificing toughness.

The dependence of brittle failure on density is also complicated by the fact that the density of any polyethylene is affected by its rate of cooling from the molten state. This effect is illustrated opposite in Table 1.

Values for densities quoted in the literature usually refer to specimens prepared in a standard way involving slow cooling. In injection moulding, however, the polyethylene is cooled rapidly and the molecular chains have no time in which to pack into their equilibrium positions and consequently the density is reduced to below the equilibrium value. Subsequently, over time, the density increases towards its equilibrium value, a process which is very slow but which is accelerated at elevated temperatures. Provided that a polyethylene is chosen with a density and MFI such that the polyethylene, when cooled at the slowest rate found in injection moulding, lies in the 'tough' region in Figure 16, no detrimental change to the mouldings impact strength will arise. But if a polyethylene in the 'brittle' region is chosen (for example, a material with a MFI of 20 g/10 min and a density greater than 0.927 g/cm^3) mouldings produced under conditions of rapid cooling will appear to be tough initially, because of the decrease in density, but may become brittle as the density increases over time.

Table 1: Effect of Cooling Rate on the Density of Polyethylene (MFI 20)

Cooling Rate	Density g/cm³		
Annealed at 140°C and cooled at 5°C per hour	0.918	0.923	0.927
Annealed at 140°C and cooled at 30°C per hour	0.916	0.921	0.925
Fast cooled in injection moulding	0.913	0.919	0.922
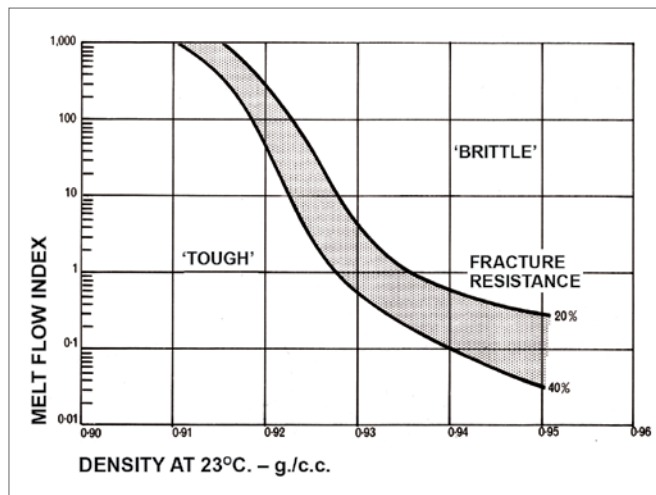
Figure 16: Variations in the "Tough Brittle" Transition (as defined by fracture resistance contours at 40% and 20%) with MFI and Density
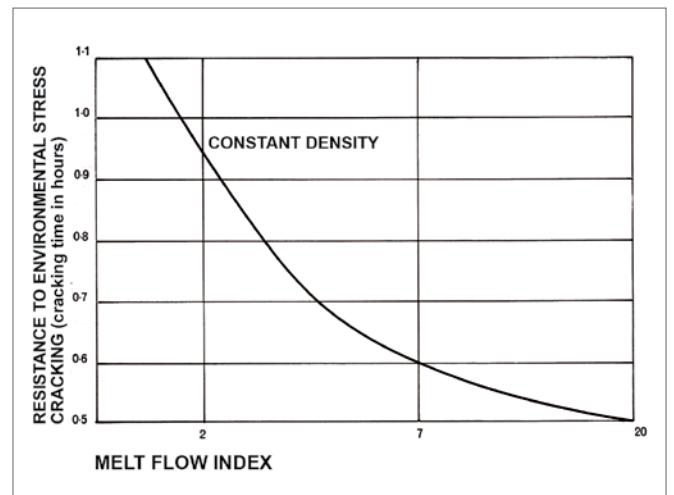
Figure 17: Resistance of Polyethylenes of Different MFI to Environment Stress Cracking

Environmental Stress Cracking

Environmental stress cracking is the name given to a phenomenon by which polyethylene under high stresses may crack in contact with certain active environments such as detergents, fats and silicone fluids.

The resistance of polyethylene to environmental stress cracking decreases rapidly as the MFI is increased. Figure 17 indicates how test specimens of polyethylenes of different MFI and of constant density behave when subjected to a severe stress in the presence of an active environment. Comparison of polyethylenes of constant MFI but of different densities is more complicated because in such tests the specimens are tested under constant strain and therefore the higher density polyethylenes will be under greater stress because they are stiffer. Nevertheless, the comparison is a valid one because in many applications, for example, screwing down a bottle closure or forcing a washing-up bowl into a sink, it is the deformation which is constant rather than the stress.

In practice it is important that high MFI polymers, even of low density, should not be used for applications in which they will be severely stressed when in contact with active environments. For such applications a polyethylene of low MFI is essential and the higher the density of the polyethylene the lower the MFI must be.

A typical application for which a polyethylene of low MFI is preferred in order to reduce the hazards of environmental stress cracking is that of closures used in contact with liquid detergents, soap solutions and certain cosmetics.

It is important however not to exaggerate the seriousness of environmental stress cracking. It has been found that the majority of mouldings made from polyethylene are not subjected to severe enough stressing in service to cause failure, even though they may be in contact with active environments. For example, most polyethylene housewares are in daily contact with both detergents and fats, and yet the externally applied stresses to which they are subjected to are not sufficient to cause failure through environmental stress cracking.

5 INJECTION MOULDING

Careful consideration needs to be made of the choice of polymer that will meet the demands of the finished product and the environment(s) that it will be exposed to (e.g. oils, fats, alkalis, acids and temperature, etc.). To make the best resin selection, customers are advised to discuss their specific end product requirements with their Qenos Technical Service Representative.

Mechanical Stress Cracking

Under certain conditions the moulding process itself can create high levels of internal stress in polyethylene. This is due to the semi-crystalline nature of the polymer which enters the mould in a molten state and undergoes crystallisation as the resin solidifies. The different polyethylenes undergo different degrees of crystallisation which is dependent on their molecular structure.

In general, the polyethylenes can be ranked in terms of their crystalisability/shrinkage in the following order:

$$HDPE \rightarrow LLDPE \rightarrow LDPE$$

The internal stress that is also commonly referred to as 'frozen in strain' or 'residual strain' may cause similar effects to those seen where polyethylene is exposed to external stresses in service.

The occurrence of 'frozen in strain' is due to both the crystalline nature of the resins used and also as a result of the moulding conditions and the design of the finished part (see Conditions for Moulding Polyethylene section on pg. 18).

Once a polyethylene has been selected (HDPE, LLDPE, LDPE) for fabrication of the finished part, internal stresses can be negated/minimised through careful mould design and by controlling the processing conditions on the injection moulding machine.

Many mouldings, however, are also subjected in service to externally applied mechanical stresses which can cause cracking. Examples of such mouldings are those containing metal inserts (e.g. knobs) and those used for interference applications (e.g. snap-on closures, ferrules or feet for tubular furniture). For such finished parts careful selection of the polymer is important. Within the polyethylenes a balance is required between the MFI (e.g. for ease of processing) and the density (e.g. which affects the level of shrinkage) in order to minimise the level of internal stress.

Generally, higher density polyethylenes would require a lower MFI and vice versa. For example, a polyethylene of MFI 20 g/10 min should generally not exceed a density of 0.918 g/cm^3. Although such "rules of thumb" are only general recommendations, other considerations of mould design and the generation of weld lines in the finished part are factors that need to be reviewed when assessing the strength of the moulding.

For articles not expected to be stressed in service, cracking caused by 'frozen-in strain' is the hazard to be avoided. A polyethylene of higher MFI is preferable because it is easier to mould such a polyethylene to give a low level of 'frozen-in strain'.

Summary

In general, polyethylenes of high MFI and low density are most commonly used for injection moulding because they give the highest outputs, have the best mould filling properties, and give the glossiest mouldings. For applications in which mouldings are likely to be stressed in service, polyethylenes of low MFI must be used. If increased stiffness is required, polyethylenes of higher density are necessary, but these must have a lower MFI to prevent them from becoming brittle and to improve resistance to environmental and mechanical stress cracking. For non-stressed applications 'frozen-in strain' is the hazard to be avoided and a polyethylene of higher MFI is preferred. Provided that these few simple principles are followed, articles giving a long and satisfactory service life can be moulded from polyethylene without difficulty.

SOME ASPECTS OF DESIGNING MOULDS FOR POLYETHYLENE

A detailed examination of mould design is outside the scope of this booklet. There are however, three problems affecting mould design which, although not peculiar to low density polyethylene, occur frequently with this material and which can conveniently be discussed here. These are:

- Shrinkage
- Distortion
- Weld lines

Shrinkage of Polyethylene Mouldings

The influence of moulding conditions and the shape of mouldings is so great that it is almost impossible to predict the exact shrinkage of polyethylene mouldings. It is recommended therefore that trials under controlled moulding conditions should be carried out before the mould is hardened and polished. The mould may then be adjusted accordingly. To allow for any after-shrinkage the dimensions of mouldings should not be checked until at least 24 hours after removing the mouldings from the mould.

Measurements must be checked in all important dimensions because mould shrinkage varies with the direction of flow, and checking only one dimension and applying proportional corrections to the others may lead to major inaccuracies.

The following major variables affect mould shrinkage.

- **Melt temperature:** the higher the melt temperature, the greater the shrinkage will be
- **Mould temperature:** the higher the mould temperature, the greater the shrinkage will be
- **Injection dwell time and injection pressure:** shrinkage will be smaller for longer injection dwell times and higher pressures
- **Thickness of section:** the thicker the moulded section, the slower the cooling and the greater the contraction of the moulding will be
- **Orientation:** shrinkage will be greater in the direction of flow than at right angles to it
- **Density:** shrinkage is greater with polyethylenes of higher density e.g. a polyethylene of density 0.930 g/cm^3 will shrink more than a polyethylene of density 0.918 g/cm^3
- **Gating:** shrinkage is usually greater when pin gates are used than when sprue gates are used

Because the above variables have such a marked effect on shrinkage, it is clear that in order to maintain accurate dimensions, close control of moulding conditions is essential. Cooling channels must provide adequate and even control of mould temperature over the whole mould. Cycle time control is of equal importance, especially for precision work. Injection pressures should be controlled and the values checked regularly on a gauge.

A point which must always be kept in mind when specifications call for close moulding tolerances is that the coefficient of thermal expansion of polyethylene is high and that a change of 5°C in room temperature will alter the length of a moulding by as much as 0.001 cm/cm.

Some examples of shrinkage are illustrated in Figure 18. Because it is usually on small mouldings that close dimensional control is required, Figure 18 shows where sink marks and warping are likely to occur in such finished items.
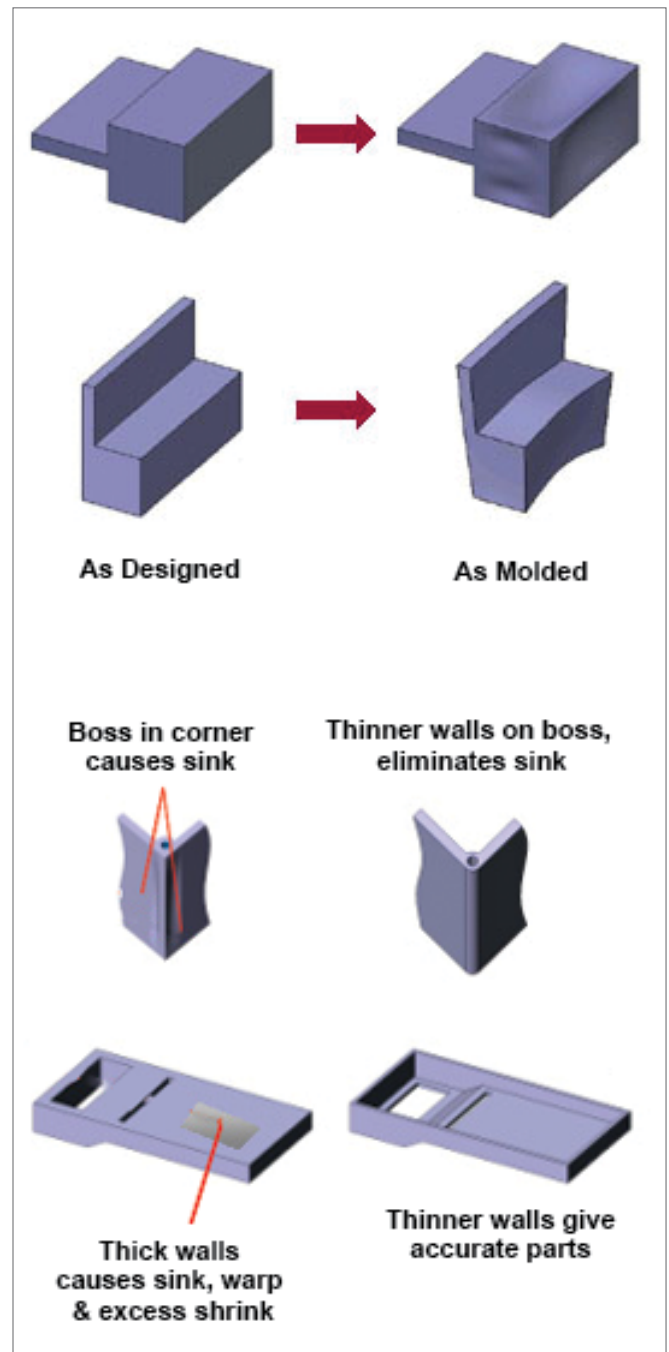
Figure 18: The Effects of Processing Conditions on Shrinkage and Warping

5 INJECTION MOULDING

Distortion of Polyethylene Mouldings

Distortion or warping of polyethylene mouldings can be a problem on flat articles which do not have a solid rim or walls to keep the base firmly held in position. The explanation of this warping is mainly due to polymer orientation and differential crystallisation across the moulding (see Figure 19).
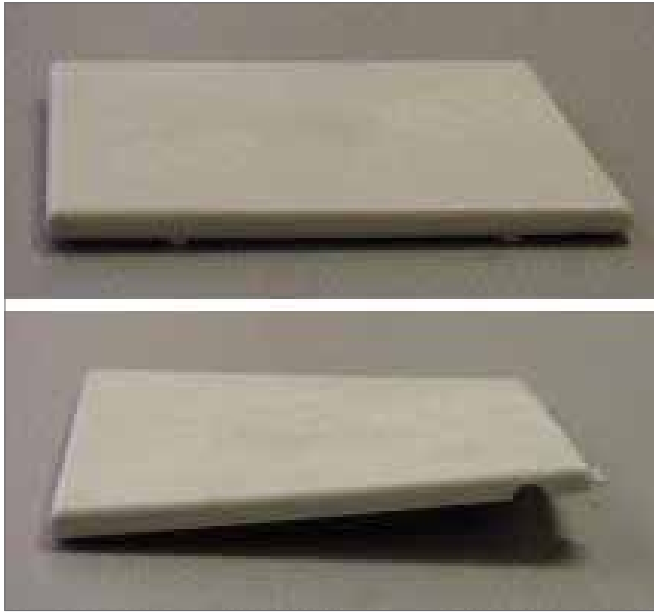
Figure 19: Processing Conditions Causing Polymer Orientation which Leads to Warping

When the mould is first filled, a hot moulding will be made. As the mould fills, the long thread-like polyethylene molecules would tend to be oriented in the direction of flow i.e. radially outwards, but as the moulding cools a radial shrinkage will occur which is greater than the shrinkage at right angles to the radius. Thus when the moulding is cold it will inevitably warp due to the difference in the stresses generated in the part. All methods of preventing the distortion of flat articles without rims or walls depend, in essence, on reducing this difference.

Mould Design

To reduce the warping in articles, multiple pin gates must be used. This system relies on reducing the length of each radial flow path and inter-mingling the melt streams, and is often adequate for low and medium density polyethylenes (see Figure 20).

Figure 20: Photos Illustrating Multiple Pin Gating and Fan Gating

For rectangular shapes the ideal gating arrangement is a fan gate (see Figure 20) all along one edge so that flow takes place mainly along the major axis. The moulding will still shrink to a greater extent in the direction of flow, causing the major axis to be proportionately shorter than the minor axis when the moulding is cold, but it will not distort. To position a gate at the end of a rectangular article is relatively easy on small mouldings to be made on multi-impression tools, but it is not so easy on large single-impression moulds. Some machine manufacturers can arrange for off-set injection points by altering the nozzle position from the usual central point and this is a very useful feature if large flat articles are to be made from high or low density polyethylene.

Choice of Polymer

The likelihood of warping increases rapidly with increasing density of the polyethylene used: high density polyethylene mouldings warp more than those of medium density, which in turn warp more than those of low density polyethylene. If flexibility in the moulding can be tolerated, a polyethylene of low density (e.g. 0.916 g/cm^3) will give the least distortion. If the mouldings are not to be stressed and physical strength is not important, e.g. sink trays and many box lids, the best results are obtained from a low density polymer of high MFI (22-70 g/10 min, according to the lack of strength which can be tolerated).

Moulding Conditions

Obviously the ideal moulding conditions would be those which give no orientation in the moulding and thus no warping. In practice such conditions can never be achieved. It has been found that long injection dwell times and high pressures, because they reduce the overall level of shrinkage, can often reduce warpage, but these conditions give rise to packing stresses and may cause the mouldings to split across the sprue. The best compromise in moulding conditions has been found to consist of a very high melt temperature (i.e. 50°C higher than that normally used for a given polyethylene) and a very cold mould (i.e. as cold as can be achieved).

Weld Lines

Weld lines can occur in any plastic moulding when the melt stream is divided as it flows round some obstruction, or can arise through non-uniform filling of the mould caused by, for example, eccentricity of cores (see Figure 21).

Figure 21: Mouldings Illustrating the Formation of Weld Lines When Two Melt Fronts Meet

Weld lines are particularly troublesome in polyethylene mouldings which are stressed in service, because failures are likely to occur some considerable time after the part has been installed. With many plastics, weld lines are immediately obvious as a physical weakness in the moulding which is detectable by brittleness on impact or flexing. With polyethylene, the fault may not appear so serious, and it may only be when stress is applied over a period of time in service, particularly in contact with an active environment, that failure will occur. Weld lines can be minimised by the use of high melt and mould temperatures, and also by utilisation of high injection pressures. Although care must be taken not to create greater difficulties by introducing packing around the sprue. A better solution however is to avoid weld line formation wherever possible by suitable positioning of the gate. On many bottle closures for example a centre pin gate can be used instead of a side gate. The mould may cost more with centre gates, but with bottle caps in particular, which are stressed in an outwards direction, the advantages of mouldings free from weld lines are great. In many cases the additional strength conferred by centre gating will permit the use of a polyethylene of high MFI which, although poorer in resistance to environmental stress cracking, will process easier and faster. Where articles of cylindrical shape are highly stressed in an outwards direction and centre gating is not possible, serious consideration should be given to diaphragm or ring gating.

5 INJECTION MOULDING

Flow Weld Lines

These generally occur towards the end of the flow path on a thin-walled article of large surface area, e.g. certain types of buckets. They are caused by the dividing of the advancing melt front into separate streams which fail to fuse together when the mould is full. This effect is aggravated by inadequate pressure on the melt or too low a melt temperature. The weld lines formed may be barely visible to the naked eye, but they can readily be detected by immersing the moulding in carbon tetrachloride at a temperature of 50 to 70°C where fissures will open up. Such weld lines are quite common and cause splits in the walls of thin containers (see Figure 22).
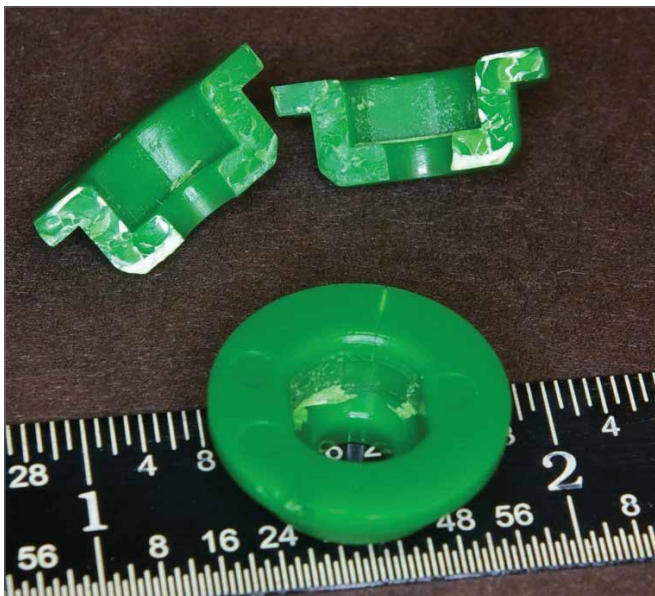

Figure 22: **Failure Due to Flow Weld Lines**

CONDITIONS FOR MOULDING POLYETHYLENE

In the injection moulding process the moulder is able to control several operating variables, each of which can influence the quality of the mouldings or the rate at which they are produced. These variables are:

- The temperature of the machine cylinder
- The temperature of the mould
- The 'injection variables', i.e. the injection pressure and speed, and the cycle time

The aim of the moulder must be to choose, for each particular material and moulding, the correct combination of variables which will produce perfect mouldings as easily and as quickly as possible. The position is somewhat complicated by the fact that a moulding that looks perfect may not in fact be so because of the presence of 'frozen-in strain', and therefore the choice of moulding conditions must take into account their effect, not only on the appearance of the moulding, but also on 'frozen-in strain'. In the following sections each variable will be discussed in the light of these two considerations, together with other relevant factors, such as the use of mould release agents. Finally a table, summarising some common moulding faults, their causes and remedies, is given (see Appendix 2).

Cylinder and Melt Temperatures

The melt temperature is the temperature of the polyethylene as it enters the mould. Depending on the grade of polyethylene being used, the temperature should lie in the range 160–280°C. In practice, it is not convenient to measure the melt temperature directly, and it is therefore necessary to use the machine cylinder temperature as a guide to the value of the melt temperature. The important point to note is that the cylinder temperature as indicated on the control panel instruments is not necessarily the same as the melt temperature, because the melt temperature depends on the rate at which the material passes through the cylinder and through the gate of the mould, as well as on the cylinder temperature. For example, if the shot weight is almost as large as the shot capacity and mouldings are being produced very rapidly, the material will be in contact with the heated cylinder for only a short time before being injected and may not have time to reach the temperature of the cylinder but may be as much as 30°C lower. On the other hand, in a machine of larger capacity that is working at slower output rates, the time of contact will be longer and consequently a lower cylinder temperature can be used and the difference between it and the melt temperature can be reduced to about 5°C. Similarly, a moulding containing a thick section will require a lower cylinder temperature than will a moulding of equal weight but of thinner section. This is because the thick moulding will require a longer cooling time and thus a longer cycle time than the thinner moulding; therefore the material will be in contact with the heated cylinder for a longer time and its temperature will more nearly approach that of the cylinder. A less common cause for the melt temperature to be different from the cylinder temperature is frictional heating of the material as it passes through the gate;

if material is injected rapidly through a small gate the heat generated may be sufficient to raise the melt temperature above that of the cylinder.

From these examples it is clear that it is not possible to predict the exact cylinder temperature that must be used to obtain a given melt temperature, but that it is necessary to choose a suitable cylinder temperature as a starting point and then to make adjustments based on visual inspection of the mouldings and on considerations of 'frozen-in strain'. For grades with MFI above 20 g/10 min the suggested starting temperature is 210°C and for grades with MFI below 20 g/10 min the suggested starting temperature is 260°C.

When the cylinder temperature has been set, the injection pressure and cycle time should be adjusted to the minimum values consistent with the production of full mouldings, and moulding should then be carried out for long enough (usually 15-30 minutes) to enable conditions to settle down. The mouldings should then be inspected and tested. Testing should be conducted after conditioning for 24 hours, preferably in a constant temperature environment.

Appearance of Mouldings

If the surface of the mouldings is dull or patchy, or contains matt rings or 'chevron marks' (see Figure 12), this is an indication that the melt temperature is too low, and the cylinder temperature should be raised until mouldings with a uniform, glossy finish are obtained. If the surface finish is acceptable, but mouldings are tending to stick in the mould, the melt temperature is probably too high and the cylinder temperature should be reduced until the trouble is eliminated. These procedures are effective for all grades of *Alkathene* LDPE but it should be remembered that with materials of MFI below 0.5 g/10 min the cycle time may have to be rather long to allow the melt to reach the required temperature.

Frozen-in Strain

Melt viscosity (and hence melt temperature) is the most important factor determining 'frozen-in strain'. As highlighted in Appendix 1 the presence of 'frozen-in strain' is associated with orientation of the polyethylene molecules as they are injected into the mould cavity. At high temperatures the viscosity of the polyethylene is low and the mould is filled rapidly: only the layer of material immediately adjacent to the mould surface has frozen before the mould is filled so that during cooling the maximum relaxation of orientation can take place.

At low moulding temperatures the melt viscosity is higher, the mould fills relatively slowly, and the polyethylene freezes quickly so that relatively little relaxation of the polymer orientation can occur. It has been shown quite conclusively, not only by laboratory tests but also by extensive service trials, that mouldings made at low melt temperatures can contain enough 'frozen-in strain' to overcome the structural integrity of the part and result in failure, whereas those made under optimum conditions are perfectly satisfactory (see Figure 22).

It may be concluded that the optimum cylinder temperature is the lowest at which full, glossy mouldings can be obtained, and that under these conditions 'frozen in strain' will be at a minimum. Too high a temperature will lead to sticking and long cycles, and too low a temperature will lead to strained mouldings.

Mould Temperature

The mould temperature chosen should be that at which good mouldings can be produced with a minimum cycle time. The colder the mould the faster the melt will cool and the greater will be the tendency for 'frozen-in strain' to develop. Therefore, to reduce 'frozen-in strain' a warm mould is recommended and for the minimum amount of strain, a heated mould (as hot as possible) would be required. However, the use of a very hot mould would slow down the cooling rate and thus not only prolong the moulding cycle but also substantially increase the density of the moulding. This is particularly true for mouldings that contain thick sections. As explained in the Impact Properties section (pg. 11), certain polyethylenes can, under these conditions, be brought from the tough region into the brittle region (see Figure 16). In practice, mould temperatures in the range 30-50°C have been found to offer the best compromise between the effects of 'frozen-in strain' and notch-sensitivity. Figure 23 shows the variation of retraction with mould temperature for a constant cylinder temperature.
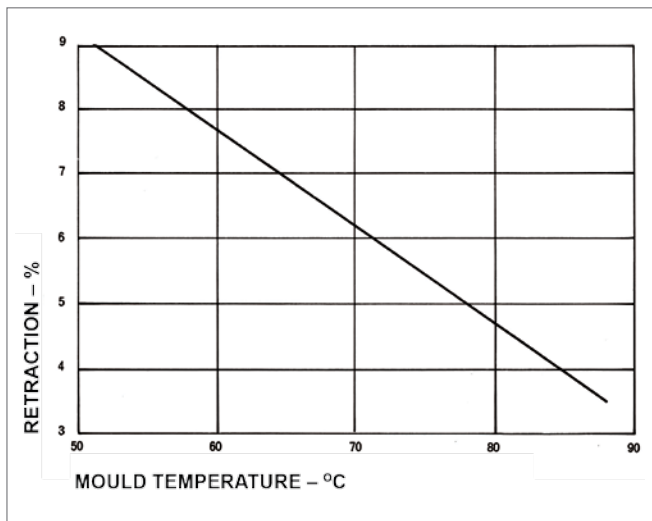
5 INJECTION MOULDING

Figure 23: Variation of Retraction with Mould Temperature (Cylinder Temperature is Constant)

Because of the importance of correct mould temperature and the growing tendency to reduce cycle times it is essential, as already remarked, that in the initial designing of the mould, provisions should be made for efficient cooling; unfortunately this is a feature which is all too often overlooked with consequent difficulties in subsequent operation.

Injection Variables

The injection variables will be considered under two headings: injection pressure and dwell time; and mould filling time.

Injection Pressure and Dwell Time

To produce good mouldings, both quickly and economically, the injection pressure should be kept to a minimum and the dwell time made as short as possible. Increasing the packing of an additional volume of polyethylene into the mould during the dwell time to compensate for the shrinkage of the polyethylene due to crystallisation is also important. The degree of packing should be kept to a minimum because the excess polyethylene is forced into the mould cavity when the melt has almost solidified and therefore orientation introduced at this stage relaxes slowly. This can result in a highly strained region being formed near the sprue/gate. The strain may be sufficient to initiate stress cracking and therefore the dwell time and injection pressure must be kept to a minimum.

Figure 24 shows mouldings made from the same type of polyethylene at the same cylinder temperature, but using different injection dwell times and pressures. The samples moulded at high pressure with a long dwell time appear indistinguishable from those moulded under more favourable conditions. But when the mouldings are cut open, it can be seen that excessively high pressures and long dwell times can result in a thickening of the base near the sprue, which in extreme cases, can result in thickness increases of approximately 30%. When the mouldings were then subjected to an accelerated service test in an active environment, the effects of too much packing constituted a very serious cracking hazard.

Mould Filling Time

On some machines the injection speed can be varied virtually independently of the injection pressure by means of a flow control valve. In long, thin flow paths the polyethylene will cool rapidly and this section will contain a fairly high degree of strain. In addition, thin-walled mouldings require higher pressures to fill the mould and, therefore, packing may occur before the extremities of the flow path have been reached. The remedy is to use a higher melt temperature and as fast an injection speed as possible. On the other hand, for thick-sectioned mouldings it is often an advantage to reduce the speed of injection so as to avoid 'jetting' and turbulence which will lead to mouldings with a poor surface finish.

Summary

The moulding conditions necessary to produce good mouldings with the best appearance and the lowest amount of 'frozen-in strain' are:

- A melt temperature just high enough to give a glossy surface to the moulding
- A mould temperature of about 30-50°C
- The minimum injection pressure and dwell time

normal injection dwell time
normal pressure

excessive injection dwell time
excessive pressure: note thickening

(a) before test

(b) After accelerated cracking test
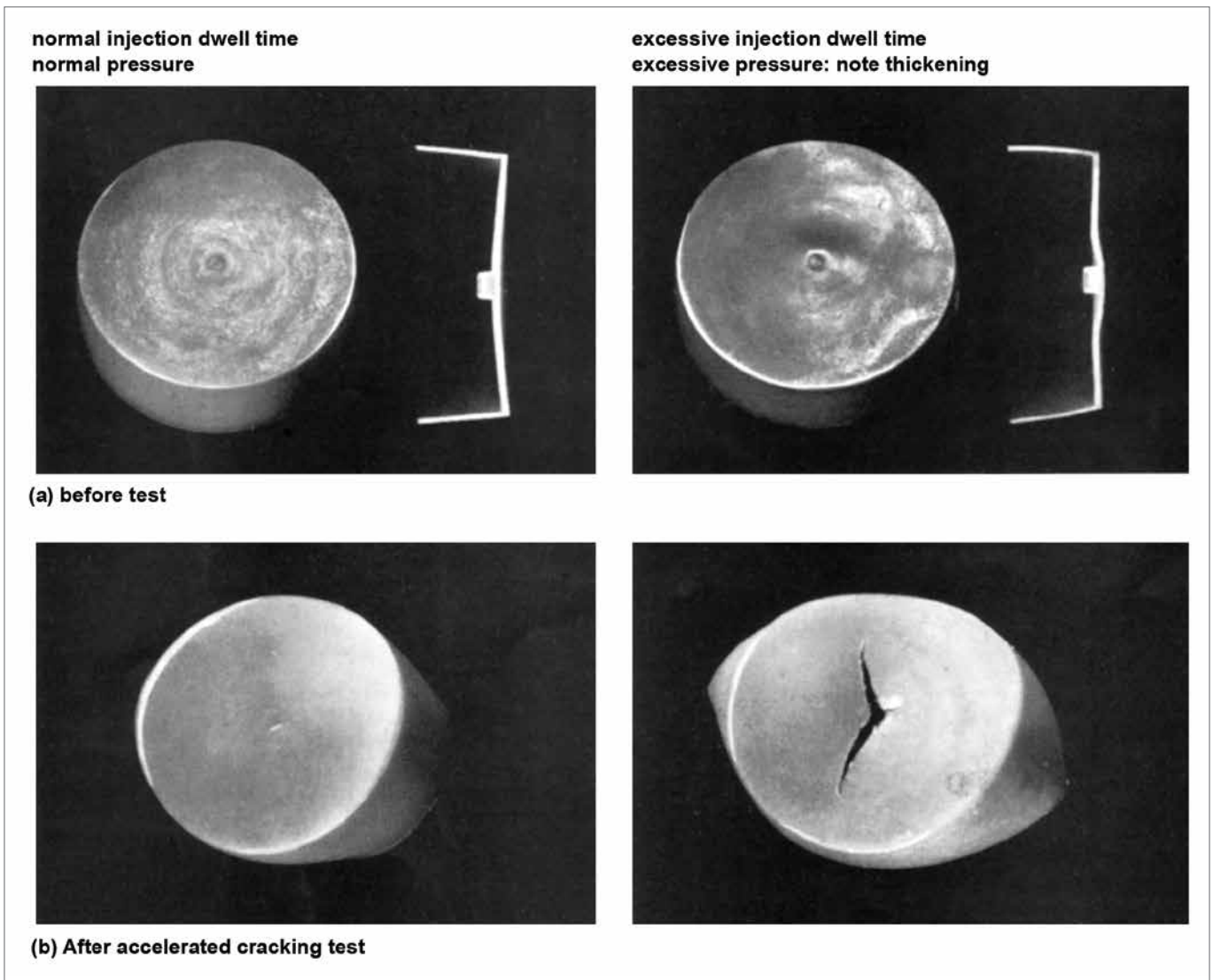
Figure 24: Effect of Injection Pressure and Dwell Time on Polyethylene Mouldings

MOULDING FAULTS

Faults in polyethylene mouldings may be divided into two classes: those that are obvious from visual inspection and those arising from the presence of 'frozen-in strain' – these can be detected only by testing. Appendix 2 lists the obvious faults that can occur, with their possible causes and remedies. Faults arising from 'frozen-in strain' have already been dealt with earlier.

In using Appendix 2 it should be noted that because the machine variables are interdependent a remedy that involves the adjustment of any one machine variable may also necessitate adjustment of the others. Alteration of the melt temperature should be gradual, in steps of 10°C, and a full cylinder of material should be injected before the results of any 10°C step are assessed. Alteration of the cycle time (which affects the length of time the material is in the cylinder and hence the melt temperature) should also be carried out gradually. Enough time should be allowed between successive adjustments to ensure that steady conditions at any one setting are obtained before the effect of that setting on the quality of the mouldings is determined.

5 INJECTION MOULDING

MOULD RELEASE AGENTS

If the correct moulding conditions have been chosen, polyethylene mouldings are unlikely to stick in the mould. If they do, and the fault cannot be corrected by adjusting the moulding conditions, mould lubricants such as stearates or fatty amides may be used. Silicone oils and greases may cause environmental stress cracking in polyethylene mouldings and therefore before they are used as mould release agents they should be tested with the moulding to see if they are suitable. If any doubt exists as to their suitability they should not be used.

DECORATING POLYETHYLENE MOULDINGS

There are several ways in which polyethylene mouldings can be decorated. These fall into two classes: those applied directly to the polyethylene surface; and those which require some form of pre-treatment of the surface. The following sections briefly deal with the various methods of pre-treatment, decoration and also with tests for the effectiveness of these processes.

Decorating Untreated Polyethylene

The following methods are commonly used:

- Hot stamping
- Labelling
- Embossing

Hot Stamping

Basically, this method consists of pressing on to the polyethylene a tape which is coated with pigment. Heat and pressure are applied via a male die and the pigment is released from the tape and fused into the polyethylene. Stamping should preferably be carried out while the moulding is still warm after being ejected from the die.

Because it is recessed, the coating obtained by hot stamping has a good degree of scratch resistance. Other advantages of this process are the absence of solvents and negating the need for drying facilities.

Labelling

Labelling is an inexpensive way of achieving a very wide range of effects. The choice of adhesive will depend on whether the label is required to be permanently fixed or easily removed.

Embossing

A relief pattern on mouldings is easily achieved by cutting the pattern in the mould. Conversely, a relief pattern on the mould produces a corresponding recessed pattern in the moulding. The embossed design can subsequently be decorated by printing or by painting. A wide range of textures and finishes can be obtained by this method.

Decorating Treated Polyethylene

Pre-treatment

Because polyethylene is non-polar and cannot be dissolved in any known solvent at room temperature it is not possible to directly apply conventional inks, paints and lacquers. There are, however, several ways in which polyethylene can be made polar. These are:

- Chlorination
- Chemical oxidation
- Flaming
- Electronic methods

Of these, chlorination is of little commercial importance, and electronic methods are usually restricted to thin films. Flaming is a versatile process which can handle any surfaces which do not contain deep or intricately shaped recesses. Chemical methods are not used so frequently, but they are the most satisfactory for parts of complex design.

Flame Treatment

Flaming a polyethylene moulding results in slight oxidation of the surface. This provides a polar surface which is required for good adhesion. The flame should be oxygen rich, of constant length and should impinge on the surface long enough to result in dulling of the surface. The exact technique will vary according to the shape of the part being treated. The essential point is that all parts of the surface should be uniformly treated.

Chemical Treatment

Chemical methods of pre-treatment involving acid etching are costly and often difficult to operate, but they are used for complicated parts and for parts to be vacuum metallised. Basically the procedure is simple:

- The moulding is immersed for 30 sec to 2 min in an acidified dichromate solution (a typical solution is 100 cm^3 of concentrated sulphuric acid, 50 cm^3 water and 15 g of potassium dichromate),
- Removed from the bath, washed thoroughly and dried.

The big disadvantage of this method is the need to handle acid solutions; the main advantage is that every part of the surface, provided it is clean, is treated in the same way.

Tests for Pre-treatment

It is obviously desirable to be able to test the effectiveness of any pre-treatment to ensure good adhesion of the finished coating. Several tests can be used, of which those based on 'wettability' of the surface are popular because of their simplicity.

Peel Test

This test involves the use of a solvent-free, pressure sensitive tape. Such a tape has little affinity for an untreated polyethylene surface and is removed fairly easily, whereas it will bond strongly to a treated surface. A suitable tape is No. 850 supplied by Minnesota Mining and Manufacturing Co. Ltd. (3M). The tape is rolled on to the moulding by means of a rubber roller and is then peeled off under standard conditions using a tensometer. By noting the 'peel strength' recorded, a quantitative indication of the treatment level can be obtained. Since decorative coatings vary in their adhesion to polyethylene surfaces, there is no basic correlation between peel strength and adhesion. However, it has been found that treatments giving peel strengths greater than about 120 g/cm will result in satisfactory adhesion of most coatings.

Decorating Methods for Treated Surfaces

Two methods that can be used are:

- Silk-screen printing
- Vacuum metalising

Silk-screening

This is essentially a stencilling process in which the stencil takes the form of a silk, nylon or metal screen which has been made porous, by a photographic process, over areas corresponding to the design to be printed. The screen is held taut in a wooden frame which also serves as a reservoir for the ink. In use, the screen, with ink on its upper surface, is placed in contact with the article and a rubber 'squeegee' is drawn over the screen, thus forcing ink through the porous area on to the article.

Screen printing has the great advantage of low capital cost, particularly when the operation is done manually. Fully automatic units are available. The main disadvantage of silk-screening is that no more than one colour can be applied at one pass. If additional colours need to be applied, then the moulding must be dried before the next colour is applied.

Vacuum Metallising

In vacuum metallising a thin continuous layer of metal is deposited onto a prepared surface by vaporising the metal under high vacuum and condensing it on the surface. In practice, a lacquer is applied to the pre-treated polyethylene as a base coat. This serves to smooth out any imperfections and also acts as a key for the metallic film. The metallic film (usually of aluminium) is deposited, and a top coat of protective lacquer is applied. Low density polyethylene articles are successfully finished in this way. Although the flexibility of the material is a disadvantage.

Tests for Finished Coatings

Two simple but effective tests are the Scratch test and the Scotch Tape test.

Scratch Test

A good idea of the adhesion of a coating can be obtained by scratching it with a finger nail or a knife to see if it flakes.

Scotch Tape Test

In this test a length of pressure-sensitive tape such as Scotch Tape supplied by 3M is stuck on to the polyethylene moulding and then pulled off, slowly at first and then more quickly. The level of adhesion of the coating can be judged qualitatively by the degree, if any, to which the coating is removed.

5 INJECTION MOULDING

APPENDIX 1 – FROZEN-IN STRAIN

It is believed that 'frozen-in strain' develops in the following way. As the polyethylene melt is injected into the mould cavity, it is subjected to high shear forces which produce a certain degree of uncoiling of the molecular chains and causes them to be oriented in the direction of flow. The nearer the melt is to the mould surface, the greater will be the shear stress and the greater the orientation. Because the material nearest to the mould surface cools more rapidly than the material in the interior, this orientation is unable to relax and becomes frozen into position. Thus a highly oriented layer is formed, the thickness of which depends on the temperatures of the melt and of the mould surface. On the other hand, the material on the inside is insulated from the cool mould by a layer of polyethylene and consequently it remains molten until near the end of the moulding cycle. Not only is this material less oriented during mould filling, but most of the orientation that does occur can relax during the cooling stage. Therefore an injection moulded section has a composite structure consisting of a skin which is highly strained and inner layers containing a much lower degree of molecular orientation. Figure 25 is a greatly magnified picture of a section cut through an injection moulding which shows clearly the different layers that are formed. In service, the oriented chains will tend to revert to their normal, coiled configuration and this tendency is reflected in a reduction in the dimensions of a specimen parallel to the direction of flow and an increase in the dimensions at right angles to the flow. If these dimensional changes are resisted by the shape of the moulding, mechanical forces arise which can produce internal stresses large enough to cause cracking in the presence of an active environment.

At elevated temperatures the tendency for the oriented molecules to revert to their normal configuration is increased and some measure of the degree of orientation can be obtained by cutting specimens from a moulding and measuring the percentage retraction which takes place in the direction of flow when the specimens are heated. A large retraction indicates a high level of 'frozen-in strain'.

If a highly strained surface comes into contact with an active environment such as synthetic detergents or fat, a small crack may develop which is likely to propagate rapidly, especially at elevated temperatures. Depending on the particular type of polyethylene, either cracks may develop throughout the whole section or failure may be restricted to surface peeling.
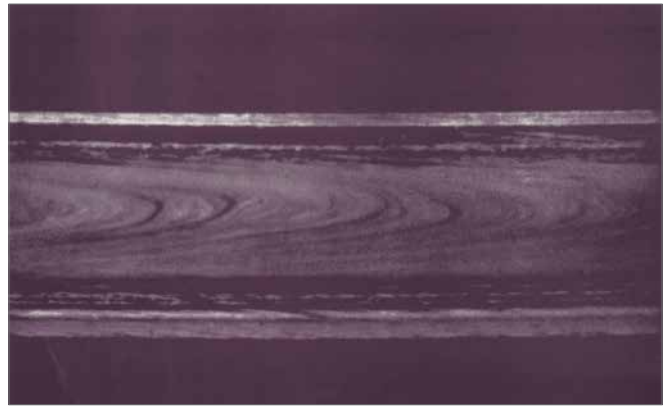

Figure 25: A Section from a Polyethylene Moulding, Showing the Layered Structure

APPENDIX 2 – INJECTION MOULDING TROUBLESHOOTING GUIDE

Problem /Issue	Cause(s)	Potential Solution(s) /Action(s)
Brittle mouldings	Sharp corners, notches	Increase radii
	PE grade has insufficient impact strength	Use lower flow and/or lower density grade of PE
	Excessive orientation	Increase melt temperature
	Inadequate thickness	Increase thickness of moulding
Burn marks. Carbonised material at end of flow path	Insufficient venting	Increase venting
	Injection speed too high	Reduce injection speed
	Melt temperature too high	Reduce barrel and nozzle temperature settings
Delamination	Incompatible masterbatch	Ensure PE based masterbatch is used
	Contaminant	Check feed for contamination
	Material freezing prematurely	Increase temperature settings. Increase gate size
Demoulding difficulties	Poor design, insufficient draft angles	Increase draft angles, incorporate "slip" additive
	Over packing	Reduce injection speed and or second stage time/ pressure, use higher flow PE grade
	Excessive second stage	Reduce second stage pressure and/or time
Distortion	Moulded in stress/ orientation	Increase melt temperature. Use increased melt flow index grade of PE
	Ribs too thick	Employ more, but thinner ribs to impart stiffness
	Variation in thickness	Use ribs for varying thickness rather than solid walls
	Variation in mould cooling	Increase cooling channels in difficult to cool areas
	Sink marks	Increase second stage pressure and or time
	Gate freezing off too quickly	Increase gate size
Flashing	Inadequate clamp force	Increase clamp force. Move mould to a higher clamp force machine
	Excessive vent size	Reduce venting
	PE melt flow index too high	Change to a low flow grade of PE
	Excessive injection speed	Reduce injection speed
Matt or streaky surface	Gate inappropriately positioned resulting in snake – like jetting	Position gate so that the material is forced to change direction immediately upon entering the mould
	Melt disturbance resulting Matt – Gloss pattern	Increase melt temperature. Reduce injection speed
	Moisture	Dry the polyethylene or masterbatch
	Incompatible masterbatch	Change masterbatch to one with a PE base
Poor colour homogenisation	Back pressure too low	Increase back pressure
	Masterbatch not compatible	Ensure PE based masterbatch is used
	Barrel size too small, insufficient shots in barrel	Move to a larger machine
	Masterbatch add rate too low	Use masterbatch with lower pigment concentration at higher add rate
	Temperature too low	Increase temperature settings

5 INJECTION MOULDING

Problem/Issue	Cause(s)	Potential Solution(s)/Action(s)
Short shots. Incompletely filled mouldings	PE melt flow index too low	Change to higher melt flow index grade
	Melt temperature too low	Increase melt temperature.
	Inadequate vent size	Increase venting
	Inadequate thickness	Increase thickness
	Insufficient injection speed	Increase injection speed
	Insufficient gating	Increase gate size or number
Weak weld lines	Melt temperature too low	Increase temperature settings
	Flow of polymer too low	Use higher melt flow grade
	Injection speed too low	Increase injection speed
	Gate(s) too far from weld line	Move gate or increase number of gates

Disclaimer

The proposed solutions in this guide are based on conditions that are typically encountered in the manufacture of products from polyethylene. Other variables or constraints may impact the ability of the user to apply these solutions. Qenos also refers the user to the disclaimer at the beginning of this document.

BIBLIOGRAPHY/FURTHER READING

1. Rosato, D. V.; Rosato, D. V.; Rosato, M. G.; Injection Moulding Handbook (3rd Ed.), Kluwer Academic Publishers, 2000.

2. Johannaber, F.; Injection Moulding Machines – A User's Guide, (4th Ed.), Hanser Verlag, 2008.

3. Bryce, D. M.; Plastic Injection Moulding – Manufacturing process fundamentals, Society of Manufacturing Engineers, 1996.

4. Osswald, T. A.; Turnig, L.; Gramann, P. J.; Injection Moulding Handbook, Hanser Verlag, 2008.

5. Potsch, G.; Michaeli, W.; Injection Moulding An Introduction, (2nd Ed.), Hanser Verlag, 2008.

6. Rueda, D. R.; Balta Calleja, F. J.; Bayer, R. K.; J. Mat Sci, 16, 3371, 1981.
    Influence of processing conditions on the structure and surface microhardness of injection-moulded polyethylene.

Issued July 2015.

Polyethylene Technical Guide Series

Qenos

01 GENERAL
PROPERTIES
–
TECHNICAL GUIDE

Alkathene™ Alkatuff® Alkamax™ Alkadyne™ Alkatane™

Qenos

02 EXTRUSION
–
TECHNICAL GUIDE

Qenos

03 FILM EXTRUSION
AND CONVERSION
–
TECHNICAL GUIDE

Alkathene™ Alkatuff® Alkamax™ Alkatane™

Qenos

04 EXTRUSION COATING
& LAMINATION
–
TECHNICAL GUIDE

Alkathene™ Alkatuff®

Qenos

05 INJECTION
MOULDING
–
TECHNICAL GUIDE

Alkathene™ Alkatuff® Alkatane™

Qenos

06 ROTATIONAL
MOULDING
–
TECHNICAL GUIDE

Alkatuff®

Qenos

07 PIPE AND TUBING
EXTRUSION
–
TECHNICAL GUIDE

Alkadyne™

Qenos

08 POLYETHYLENE
BLOW MOULDING
–
TECHNICAL GUIDE

Alkatane™

Qenos

Certified System
Quality
ISO 9001
SAI GLOBAL

AUSTRALIAN MADE