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.
# TABLE OF CONTENTS

**INRODUCTION** 6

**EFFECT OF TYPE OF POLYETHYLENE ON PROCESSING AND PROPERTIES OF MOULDINGS** 6

- Classification of Polyethylenes 6
- MFI 6

**DENSITY** 7

- Effect of MFI and Density on Moulding Characteristics 7

**MOULD FILLING** 8

- Surface Finish 9
- Summary 11

**EFFECT OF MFI AND DENSITY ON THE PROPERTIES OF POLYETHYLENE MOULDINGS** 11

- Stiffness 11
- Impact Properties 11
- Environmental Stress Cracking 13
- Mechanical Stress Cracking 14
- Summary 14

**SOME ASPECTS OF DESIGNING MOULDS FOR POLYETHYLENE** 14

- Shrinkage of Polyethylene Mouldings 14
- Distortion of Polyethylene Mouldings 16
- Mould Design 16
- Choice of Polymer 17
- Moulding Conditions 17
- Weld Lines 17
- Flow Weld Lines 18

**CONDITIONS FOR MOULDING POLYETHYLENE** 18

- Cylinder and Melt Temperatures 18
- Appearance of Mouldings 19
- Frozen-in Strain 19
- Mould Temperature 19
- Injection Variables 20
- Injection Pressure and Dwell Time 20
- Mould Filling Time 20
- Summary 20
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>MOULDING FAULTS</td>
<td>21</td>
</tr>
<tr>
<td>MOULD RELEASE AGENTS</td>
<td>22</td>
</tr>
<tr>
<td>DECORATING POLYETHYLENE MOULDINGS</td>
<td>22</td>
</tr>
<tr>
<td>Decorating Untreated Polyethylene</td>
<td>22</td>
</tr>
<tr>
<td>Hot Stamping</td>
<td>22</td>
</tr>
<tr>
<td>Labelling</td>
<td>22</td>
</tr>
<tr>
<td>Embossing</td>
<td>22</td>
</tr>
<tr>
<td>Decorating Treated Polyethylene</td>
<td>22</td>
</tr>
<tr>
<td>Pre-treatment</td>
<td>22</td>
</tr>
<tr>
<td>Flame Treatment</td>
<td>22</td>
</tr>
<tr>
<td>Chemical Treatment</td>
<td>22</td>
</tr>
<tr>
<td>Tests for Pre-treatment</td>
<td>23</td>
</tr>
<tr>
<td>Peel Test</td>
<td>23</td>
</tr>
<tr>
<td>Decorating Methods for Treated Surfaces</td>
<td>23</td>
</tr>
<tr>
<td>Silk-screening</td>
<td>23</td>
</tr>
<tr>
<td>Vacuum Metallising</td>
<td>23</td>
</tr>
<tr>
<td>Tests for Finished Coatings</td>
<td>23</td>
</tr>
<tr>
<td>Scratch Test</td>
<td>23</td>
</tr>
<tr>
<td>Scotch Tape Test</td>
<td>23</td>
</tr>
<tr>
<td>APPENDIX 1 – FROZEN-IN STRAIN</td>
<td>24</td>
</tr>
<tr>
<td>APPENDIX 2 – INJECTION MOULDING TROUBLESHOOTING GUIDE</td>
<td>25</td>
</tr>
<tr>
<td>BIBLIOGRAPHY/FURTHER READING</td>
<td>27</td>
</tr>
</tbody>
</table>
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.

Disclaimer

All information contained in this publication and any further information, advice, recommendation or assistance given by Qenos either orally or in writing in relation to the contents of this publication is given in good faith and is believed by Qenos to be as accurate and up-to-date as possible.

The information is offered solely for your information and is not all-inclusive. The user should conduct its own investigations and satisfy itself as to whether the information is relevant to the user’s requirements. The user should not rely upon the information in any way. The information shall not be construed as representations of any outcome. Qenos expressly disclaims liability for any loss, damage, or injury (including any loss arising out of negligence) directly or indirectly suffered or incurred as a result of or related to anyone using or relying on any of the information, except to the extent Qenos is unable to exclude such liability under any relevant legislation. Freedom from patent rights must not be assumed.
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).

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.
DENSITY

Density is related to the crystallinity of the polyethylene and is measured in g/cm³.

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 crystalites. 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.

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.
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

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.
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.

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.
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).
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/cm³ will double the stiffness. Figure 14 also shows the effect of temperature on stiffness.

MFI has virtually no effect on stiffness.

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.
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/cm³) 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.
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.
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 crystallisability/shrinkage in the following order:

\[
\text{HDPE} \rightarrow \text{LLDPE} \rightarrow \text{LDPE}
\]

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/cm³. 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/cm³ will shrink more than a polyethylene of density 0.918 g/cm³
- **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
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).

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).
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/cm³) 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.
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
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.
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
**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.
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 cm³ of concentrated sulphuric acid, 50 cm³ 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.
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.
# APPENDIX 2 – INJECTION MOULDING TROUBLESHOOTING GUIDE

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

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

**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

   Influence of processing conditions on the structure and surface microhardness of injection-moulded polyethylene.

Issued July 2015.