

A Guide to Polyolefin Injection Molding



lyondellbasell

A Guide to Polyolefin Injection Molding

Table of Contents

Introduction	2
Polyolefins are derived from petrochemicals	2
Molecular structure and composition affect properties and processability	2
Chain branching	3
Density	3
Molecular weight	4
Molecular weight distribution	4
Copolymers	5
Modifiers and additives	5
Working closely with molders	5
How polyolefins are made	5
Low density polyethylene (LDPE)	6
High density polyethylene (HDPE)	6
Linear low density polyethylene (LLDPE)	7
Polypropylene	7
Shipping and handling polyolefin resins	7
Material handling	7
How to solve material handling problems	9
Other material handling practices	10
The injection molding process	11
Injection units	11
Plasticator specifications	13
Screw designs	13
Nozzles	14
Clamp mechanisms	15
Clamp specifications	16
Injection molds	17
Types of mold	17
Sprues and runners	17
Mold venting	19
Gating	20
Mold cooling	20
Ejection devices	21
Spiral flow measurement	21
General injection molding operating procedures	22
General safety	22
Heat	23
Electricity	23
Machinery motion	23
The injection molding process and its effect on part performance	23
The molding cycle	23
Shrinkage	29
Warping	29
Color dispersion and air entrapment	30
Part ejection and mold release	31
Clarity	32
Gloss	32
Polypropylene integral hinges	32
Appendices	
1. Injection Molding Terms	33
2. Metric Conversion Guide	38
3. Abbreviations	40
4. ASTM test methods applicable to polyolefins	41
5. Injection molding problems, causes and solutions	42
6. ASTM and ISO sample preparation and test procedures	46
7. Compression and injection molded sample preparation for HDPE	47

A Guide To Polyolefin Injection Molding

Introduction

Polyolefins are the most widely used plastics for injection molding. This manual, **A Guide to Polyolefin Injection Molding**, contains general information concerning materials, methods and equipment for producing high quality, injection molded, polyolefin products at optimum production rates.

Polyolefins that can be injection molded include:

- Low density polyethylene (LDPE)
- Linear low density polyethylene (LLDPE)
- High density polyethylene (HDPE)
- Ethylene copolymers, such as ethylene vinyl acetate (EVA)
- Polypropylene and propylene copolymers (PP)
- Thermoplastic olefins (TPO)

In general, the advantages of injection molded polyolefins compared with other plastics are:

- Lightweight
- Outstanding chemical resistance
- Good toughness at lower temperatures
- Excellent dielectric properties
- Non-hygroscopic

The basic properties of polyolefins can be modified with a broad range of fillers, reinforcements and chemical modifiers. Furthermore, polyolefins are considered to be relatively easy to injection mold.

Major application areas for polyolefin injection molding are:

- Appliances
- Automotive products
- Consumer products

- Furniture
- Housewares
- Industrial containers
- Materials handling equipment
- Packaging
- Sporting goods
- Toys and novelties

This manual contains extensive information on the injection molding of polyolefins. However, it makes no specific recommendations for the processing of LyondellBasell resins for specific applications. For more detailed information please contact your LyondellBasell polyolefins sales or technical service representative.

Polyolefins are derived from petrochemicals

Polyolefins are plastic resins polymerized from petroleum-based gases. The two principal gases are ethylene and propylene. Ethylene is the principal raw material for making polyethylene (PE and ethylene copolymer resins; propylene is the main ingredient for making polypropylene (PP) and propylene copolymer resins. Polyolefin resins are classified as thermoplastics, which means that they can be melted, solidified and melted again. This contrasts with thermoset resins, such as phenolics, which, once solidified, can not be reprocessed.

Most polyolefin resins for injection molding are used in pellet form.

The pellets are about 1/8 inch long and 1/8 inch in diameter and usually somewhat translucent to white in color. Many polyolefin resins contain additives, such as thermal stabilizers. They also can be

compounded with colorants, flame retardants, blowing agents, fillers, reinforcements, and other functional additives such as antistatic agents and lubricants.

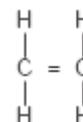
Molecular structure and composition affect properties and processability

Four basic molecular properties affect most of the resin characteristics essential to injection molding high quality polyolefin parts. These molecular properties are:

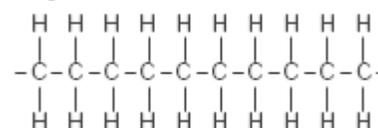
- Chain branching
- Crystallinity or density
- Average molecular weight
- Molecular weight distribution

The materials and processes used to produce the polyolefins determine these molecular properties.

The basic building blocks for the gases from which polyolefins are derived are hydrogen and carbon atoms. For polyethylene, these atoms are combined to form the ethylene monomer, C₂H₄.

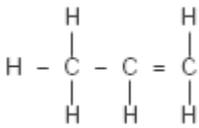


In the polymerization process, the double bond connecting the carbon atoms is broken. Under the right conditions, these bonds reform with other ethylene molecules to form long molecular chains.

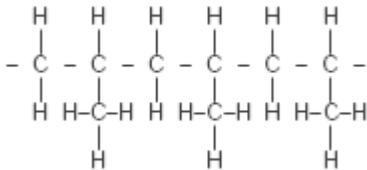


The resulting product is polyethylene resin.

For polypropylene, the hydrogen and carbon atoms are combined to form the propylene monomer, $\text{CH}_3\text{CH}:\text{CH}_2$.



The third carbon atom forms a side branch which causes the backbone chain to take on a spiral shape.



Ethylene copolymers, such as ethylene vinyl acetate (EVA), are made by the polymerization of ethylene units with randomly distributed vinyl acetate (VA) comonomer groups.

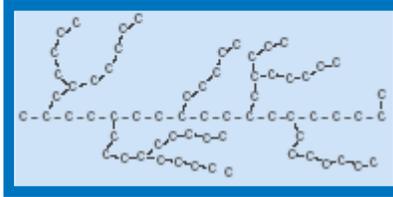
Chain branching

Polymer chains may be fairly linear, as in high density polyethylene, or highly branched as in low density polyethylene. For every 100-ethylene units in the polyethylene molecular chain, there can be one to ten short or long branches that radiate three-dimensionally (Figure 1). The degree and type of branching are controlled by the process (reactor), catalyst, and/or any comonomers used.

Chain branching affects many of the properties of polyethylenes including density, hardness, flexibility and transparency, to name a few. Chain branches also become points in the molecular structure where oxidation may occur. If excessively high temperatures are reached during processing, oxidation can occur which may adversely affect the polymer's properties. This oxidation

or degradation may cause cross-linking in polyethylenes and chain scission in polypropylenes.

Figure 1. Polyethylene chain with long side branches



Polypropylene, on the other hand, can be described as being linear (no branching) or very highly branched. Although the suspended carbon forms a short branch on every repeat unit, it is also responsible for the unique spiral and linear configuration of the polypropylene molecule.

Density

Polyolefins are semi-crystalline polymers which means they are composed of molecules which are arranged in a very orderly (crystalline) structure and molecules which are randomly oriented (amorphous). This mixture of crystalline and amorphous regions (Figure 2) is essential in providing the desired properties to injection molded parts. A totally amorphous polyolefin would be grease-like and have poor physical properties. A totally crystalline polyolefin would be very hard and brittle.

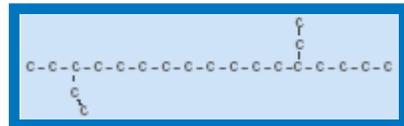
HDPE resins have linear molecular chains with comparatively few side chain branches. Therefore, the chains are packed more closely together (Figure 3). The result is crystallinity up to 95 percent. LDPE resins generally have crystallinity from 60 percent to 75 percent. LLDPE resins have crystallinity from 60 percent to 85 percent. PP resins are highly crystalline, but they are not very dense. PP resins have a nominal specific gravity

range of 0.895 to 0.905 g/cm^3 , which is the lowest for a commodity thermo-plastic and does not vary appreciably from manufacturer to manufacturer.

Figure 2. Crystalline (A) and amorphous (B) regions in polyolefin



Figure 3. Linear polyethylene chain with short side branches



For polyethylene, the density and crystallinity are directly related, the higher the degree of crystallinity, the higher the resin density. Higher density, in turn, influences numerous properties. As density increases, heat softening point, resistance to gas and moisture vapor permeation and stiffness increase. However, increased density generally results in a reduction of stress cracking resistance and low temperature toughness.

- LDPE resins have densities ranging from 0.910 to 0.930 grams per cubic centimeter (g/cm^3)
- LLDPE resins range from 0.915 to 0.940 g/cm^3
- HDPE resins range from >0.940 to >0.960 g/cm^3

As can be seen, all natural polyolefin resins, i.e, those without any fillers or reinforcements, have densities less than 1.00 g/cm^3 . This light weight is one of the key

advantages for parts injection molded from polyolefins. A general guide to the effects of density on the properties for various types of polyethylene resins is shown in Table 1.

Molecular weight

Atoms of different elements, such as carbon, hydrogen, etc., have different atomic weights. For carbon, the atomic weight is 12 and for hydrogen it is one. Thus, the molecular weight of the ethylene unit is the sum of the weight of its six atoms (two carbon atoms x 12 + four hydrogen x 1) or 28.

Unlike simple compounds, like ethylene or propylene, every polyolefin resin consists of a mixture of large and small chains, i.e., chains of high and low molecular weights. The molecular weight of the polymer chain generally is in the thousands and may go up to over one million. The average of these is called, quite appropriately, the average molecular weight.

As average molecular weight increases, resin toughness increases. The same holds true for tensile strength and environmental stress crack resistance (ESCR) – cracking brought on when molded parts are subjected to stresses in the presence of materials such as solvents, oils, detergents, etc. However, higher molecular weight results in an increase in melt viscosity and greater resistance to flow making injection molding more difficult as the average molecular weight increases.

Melt flow rate (MFR) is a simple measure of a polymer's melt viscosity under standard conditions of temperature and static load (pressure). For polyethylenes, it is often referred to as melt index (MI). MFR is the weight in grams of a melted resin that flows through a standard-sized orifice in 10

Table 1. General guide to the effects of polyethylene physical properties on properties and processing

	AS MELT INDEX INCREASES	AS DENSITY INCREASES
Durometer hardness (surface)	remains the same	increases
Gloss	improves	improves
Heat resistance (softening point)	remains the same	improves
Stress crack resistance	decreases	decreases
Mechanical flex life	decreases	decreases
Processability (less pressure to mold)	improves	remains the same
Mold shrinkage	decreases	increases
Molding speed (faster solidification)	remains the same	increases
Permeability resistance	remains the same	improves
Stiffness	remains the same	increases
Toughness	decreases	decreases
Transparency	remains the same	decreases
Warpage	decreases	increases

minutes (g/10 min). Melt flow rate is inversely related to the resin's average molecular weight: as the average molecular weight increases, MFR decreases and vice versa.

Melt viscosity, or the resistance of a resin to flow, is an extremely important property since it affects the flow of the molten polymer filling a mold cavity. Polyolefins with higher melt flow rates require lower injection molding processing pressures, temperatures and shorter molding cycles (less time needed for part cooling prior to ejection from the mold). Resins with high viscosities and, therefore, lower melt indices, require the opposite conditions for injection molding.

It should be remembered that pressure influences flow properties. Two resins may have the same melt index, but different high-pressure flow properties. Therefore, MFR or MI must be used in conjunction with other characteristics, such as molecular

weight distribution, to measure the flow and other properties of resins. Generally, injection molding resins are characterized as having medium, high or very high flow.

For injection molding grades, the MFR (MI) values for polyethylenes are generally determined at 190°C (374°F) using a static load of 2,160 g. MFR values for polypropylenes are determined at the same load but at a higher temperature 230°C (446°F). The MFR of other thermoplastics may be determined using different combinations of temperatures and static load. For this reason, the accurate prediction of the relative processability of different materials using MFR data is not possible.

Molecular weight distribution

During polymerization, a mixture of molecular chains of widely varying lengths is produced. Some may be short; others may be extremely long containing several thousand monomer units.

The relative distribution of large, medium and small molecular chains in the polyolefin resin is important to its properties. When the distribution is made up of chains close to the average length, the resin is said to have a “narrow molecular weight distribution.” Polyolefins with “broad molecular weight distribution” are resins with a wider variety of chain lengths. In general, resins with narrow molecular weight distributions have good low-temperature impact strength and low warpage. Resins with broad molecular weight distributions generally have greater stress cracking resistance and greater ease of processing (Figure 4).

The type of catalyst and the polymerization process used to produce a polyolefin determines its molecular weight distribution. The molecular weight distribution (MWD) of PP resins can also be altered during production by controlled rheology additives that selectively fracture long PP molecular chains. This results in a narrower molecular weight distribution and a higher melt flow rate.

Copolymers

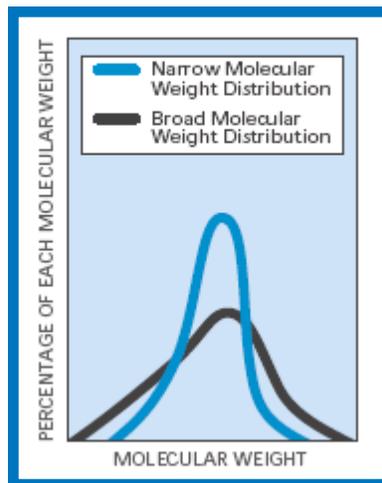
Polyolefins made with one basic type of monomer are called homopolymers. There are, however, many polyolefins, called copolymers, that are made of two or more monomers. Many injection molding grades of LLDPE, LDPE, HDPE and PP are made with comonomers that are used to provide specific property improvements.

The comonomers most often used with LLDPE and HDPE are called alpha olefins. They include butene, hexene and octene. Other comonomers used with ethylene to make injection molding grades are ethyl acrylate to make the

copolymer ethylene ethyl acrylate (EEA) and vinyl acetate to produce ethylene vinyl acetate (EVA).

Ethylene is used as a comonomer with propylene to produce polypropylene random copolymers. Polypropylene can be made more impact resistant by producing a high ethylene-propylene copolymer in a second reactor forming a finely dispersed secondary phase of ethylene-propylene rubber. Products made in this manner are commonly referred to as impact copolymers.

Figure 4. Schematic representation of molecular weight distribution



Modifiers and additives

Numerous chemical modifiers and additives may be compounded with polyolefin injection molding resins. In some grades, the chemical modifiers are added during resin manufacture. Some of these additives include:

- Antioxidants
- Acid scavengers
- Process stabilizers
- Anti-static agents
- Mold release additives
- Ultraviolet (UV) light stabilizers

- Nucleators
- Clarifiers
- Lubricants

Working closely with molders

LyondellBasell offers a wide range of polyolefin resins for injection molding, including Alathon® and Petrothene® HDPE, Petrothene® LDPE, LLDPE, and PP, Ultrathene® EVA copolymers and Flexathene® TPOs. These resins are tailored to meet the requirements of many areas of application.

Polyolefin resins with distinctly different properties can be made by controlling the four basic molecular properties during resin production and by the use of modifiers and additives. Injection molders can work closely with their LyondellBasell polyolefins sales or technical service representative to determine the resin that best meets their needs.

LyondellBasell polyolefins technical service representatives are also available to assist injection molders and end-users by providing guidance for tool and part design and the development of specialty products to fulfill the requirements of new, demanding applications.

How polyolefins are made

High-purity ethylene and propylene gases are the basic feedstocks for making polyolefins (Figure 5). These gases can be petroleum refinery by-products or they can be extracted from an ethane/propane liquified gas mix coming through pipelines from a gas field. High efficiency in the ethane/propane cracking and purification results in very pure ethylene and propylene, which are critical in the production of high quality polyolefins.

Figure 5. Olefin manufacturing process

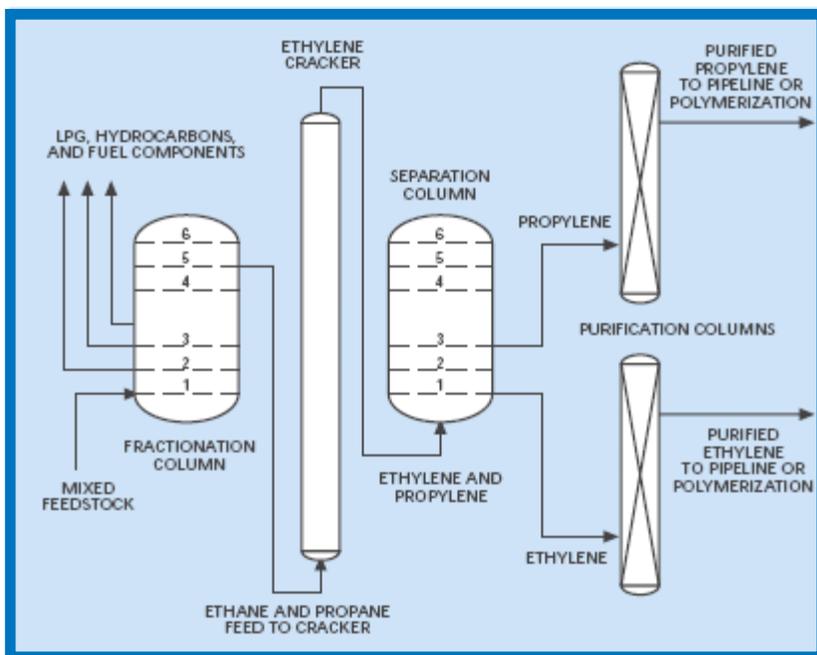
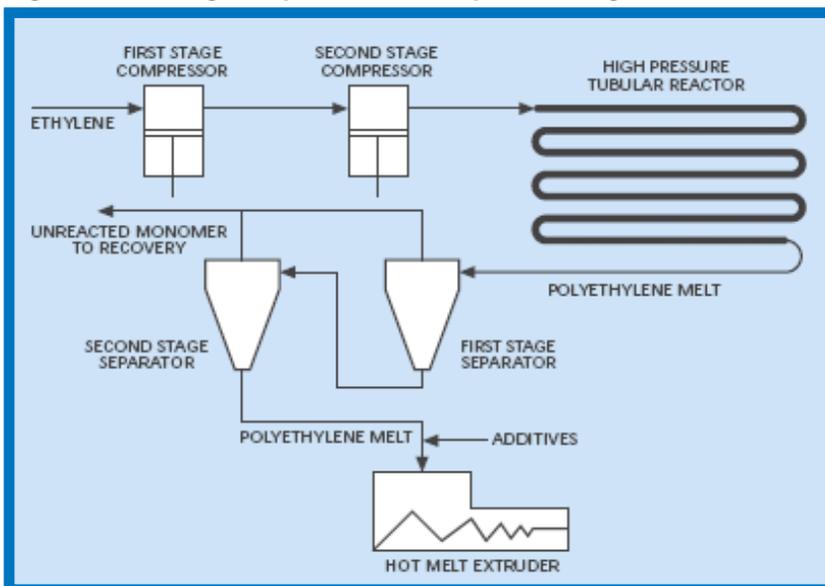


Figure 6. Left, polypropylene unit at Morris, Illinois plant. Right, HDPE unit at Matagorda, Texas plant



Figure 7. LDPE high temperature tubular process diagram



LyondellBasell can produce polyolefins by more polymerization technologies and with a greater range of catalysts than any other supplier can. Two of LyondellBasell's plants are pictured in Figure 6.

Low density polyethylene (LDPE)

To make LDPE resins, LyondellBasell uses high pressure, high temperature tubular and autoclave polymerization reactors (Figures 7 and 8). Ethylene is pumped into the reactors and combined with a catalyst or initiator to make LDPE. The LDPE melt formed flows to a separator where unused gas is removed, recovered, and recycled back into the process. The LDPE is then fed to an extruder for pelletization. Additives, if required for specific applications, are incorporated at this point.

High density polyethylene (HDPE)

There are a number of basic processes used by LyondellBasell for making HDPE for injection molding applications—including the solution process and the slurry process. In the multi-reactor slurry process used by LyondellBasell (Figure 9), ethylene and a comonomer (if used), together with an inert hydrocarbon carrier, are pumped into reactors where they are combined with a catalyst. However, in contrast to LDPE production, relatively low pressures and temperatures are used to produce HDPE. The granular polymer leaves the reactor system in a liquid slurry and is separated and dried. It is then conveyed to an extruder where additives are incorporated prior to pelletizing.

Figure 8. High temperature autoclave process diagram

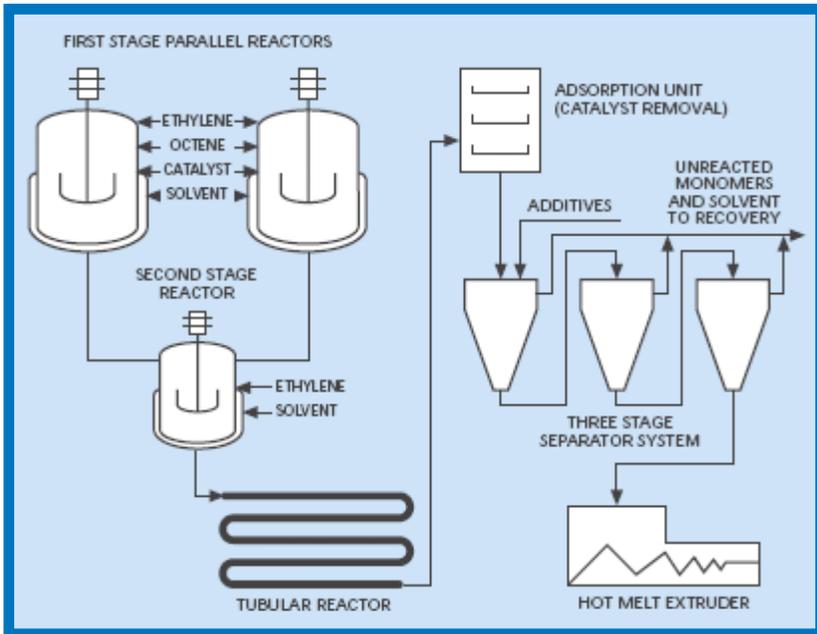
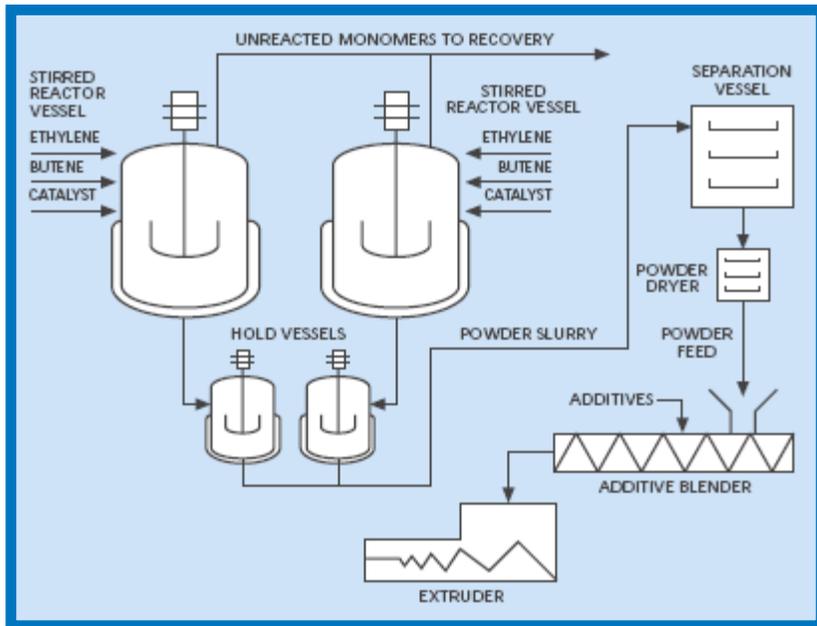


Figure 9. HDPE parallel reactors — slurry process



LyondellBasell also utilizes a multi-reactor solution process for the production of HDPE (Figure 10). In this process, the HDPE formed is dissolved in the solvent carrier and then precipitated in a downstream process. An additional adsorption step results in a very clean product with virtually no catalyst residues.

Because both of these processes utilize multiple reactors, LyondellBasell has the capability of tailoring and optimizing the molecular weight distribution of the various product grades to provide a unique range of processability and physical properties.

Linear low density polyethylene (LLDPE)

LyondellBasell uses a gas phase process for making LLDPE (Figure 11). This process is quite different from the LDPE process, but somewhat similar to the HDPE process. The major differences from the LDPE process are that relatively low pressure and low temperature polymerization reactors are used. Another difference is that the ethylene is copolymerized with butene or hexene comonomers in the reactor. Unlike HDPE, the polymer exits the reactor in a dry granular form, which is subsequently compounded with additives in an extruder.

With changes in catalysts and operating conditions, HDPE resins also can be produced in some of these LLDPE reactors.

Polypropylene

To make PP, LyondellBasell uses a vertical, stirred, fluidized-bed, gas-phase process (Figure 12). LyondellBasell was the first polypropylene supplier in the United States to use gas-phase technology to produce PP. Impact copolymers are produced using two, fluidized bed, gas phase reactors operating in series.

LyondellBasell's polyolefin production facilities are described in

Shipping and handling of polyolefin resins

It is of utmost importance to keep polyolefin resins clean. LyondellBasell ships polyolefin resins to molders in hopper cars, hopper trucks, corrugated boxes, and 50-pound plastic bags. Strict quality control throughout resin manufacture and subsequent handling, right through delivery to the molder, ensures the cleanliness of the products.

Figure 10. HDPE solution process

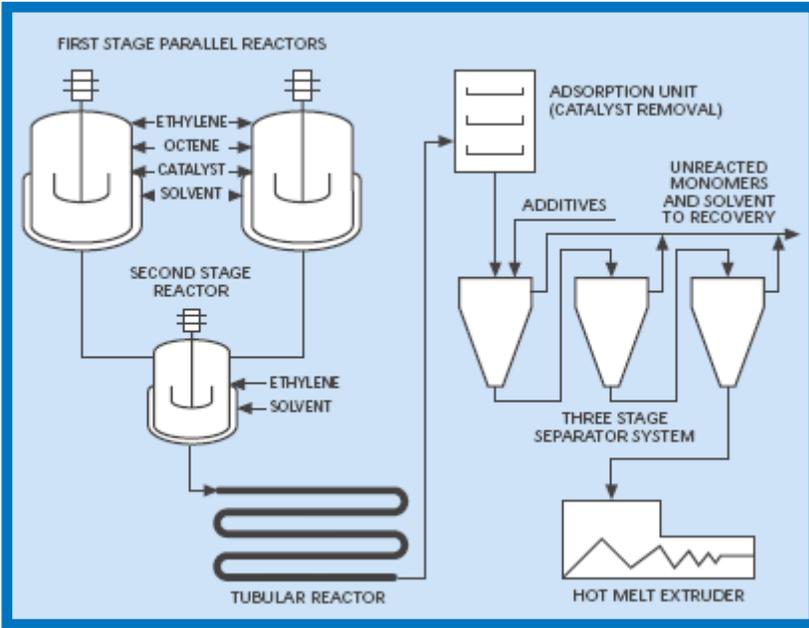
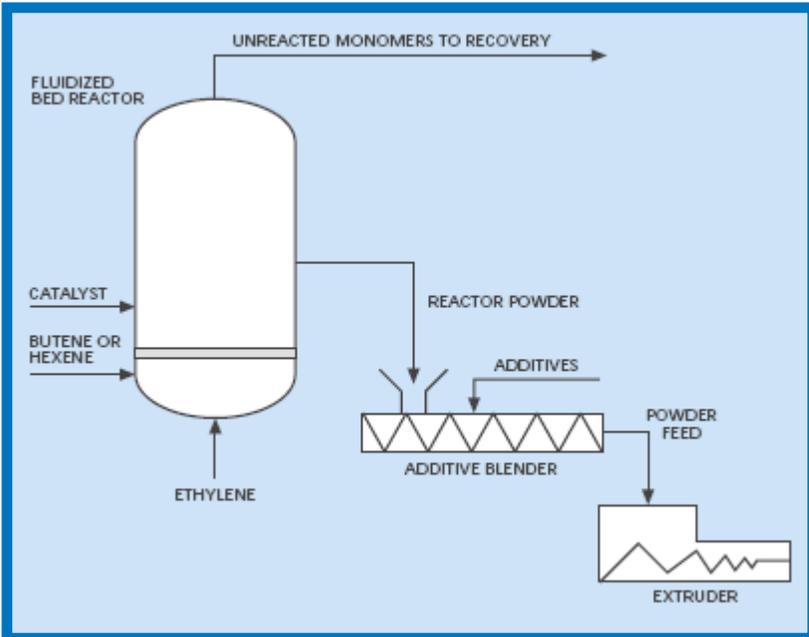


Figure 11. LLDPE fluidized bed process



When bulk containers are delivered, the molder must use appropriate procedures for unloading the resin. Maintenance of the in-plant material handling system is also essential. When bags and boxes are used, special care is necessary in opening the containers, as well as covering them, as they are unloaded.

Reground resin, whether used as a blend or as is, should also be stringently protected to keep it free of contamination. Whenever possible, the regrind material should be used as it is generated. When this is not possible, the scrap should be collected in a closed system and recycled with the same precautions taken for

Table 2. LYB polyolefin production facilities

BAYPORT, TX	Low Density Polyethylene
CHOCOLATE BAYOU, TX	High Density Polyethylene
CLINTON, IA	Low Density Polyethylene High Density Polyethylene
LaPORTE, TX	Low Density Polyethylene Linear Low Density Polyethylene
MATAGORDA, TX	High Density Polyethylene
MORRIS, IL	Low Density Polyethylene Linear Low Density Polyethylene Polypropylene
VICTORIA, TX	High Density Polyethylene

virgin resin. In all cases, the proportion of regrind used should be carefully controlled to assure consistency of processing and part performance.

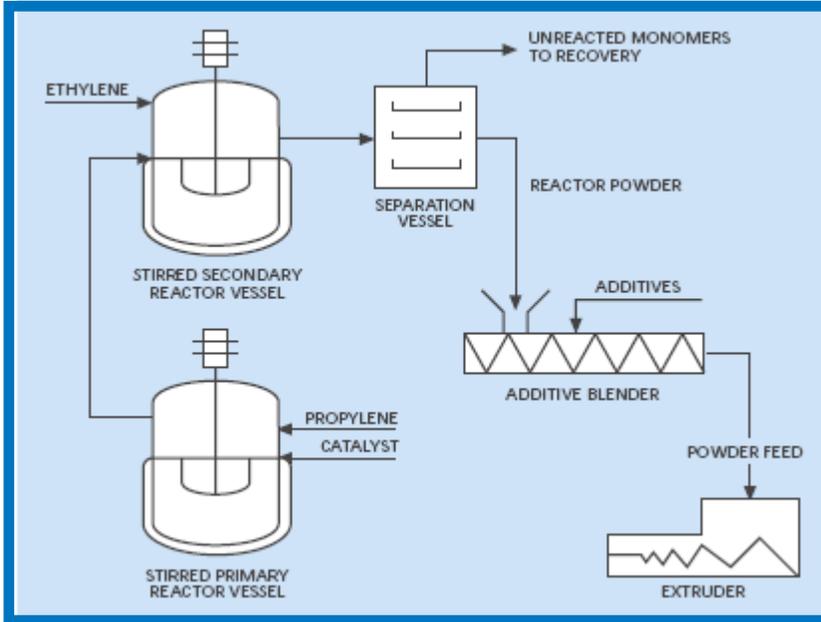
Material handling

LyondellBasell utilizes material handling systems and inspection procedures that are designed to prevent external contamination and product cross-contamination during production, storage, loading and shipment.

Since polyolefin resins are non-hygroscopic (do not absorb water) they do not require drying prior to being molded. However, under certain conditions, condensation may form on the pellet surfaces.

When cartons of resin are moved from a cold warehouse environment to a warm molding area or when transferring cold pellets from a silo to an indoor storage system, the temperature of the material should be allowed to equilibrate, for up to eight hours to drive off any condensation before molding.

Figure 12. PP dual reactors – gas-phase process



The best way to improve resin utilization is to eliminate contaminants from transfer systems. If bulk handling systems are not dedicated to one material or are not adequately purged, there is always the possibility of contamination resulting from remnants of materials previously transferred.

Occasionally, clumps of “angel hair” or “streamers” may accumulate in a silo and plug the exit port. Contaminants of this type can also cause plugging of transfer system filters and/or problems that affect the molding machine. All of these problems can result in molding machine downtime, excessive scrap and the time and costs of cleaning silos, transfer lines and filters. Polyolefin dust, fines, streamers and angel hair contamination may be generated during the transfer of polymer through smoothbore piping. These transfer systems also may contain long radius bends to convey the resin from a hopper car to the silo or holding bin. A polyolefin pellet

conveyed through a transfer line travels at a very high velocity. As the pellet contacts the smooth pipe wall, it slides and friction is generated. The friction, in turn, creates sufficient heat to raise the temperature of the pellet surface to the resin’s softening point. As this happens, a small amount of molten polyolefin is deposited on the pipe wall and freezes almost instantly. Over time, this results in deposits described as angel hair or streamers.

As the pellets meet the pipe wall, along the interior surface of a long radius bend, the deposits become almost continuous and streamers are formed. Eventually, the angel hair and streamers are dislodged from the pipe wall and find their way into the molding process, the storage silo or the transfer filters. The amount of streamers formed increases with increased transfer air temperature and velocity.

Other good practices of material handling include control (cooling) of the transfer air temperature to minimize softening and melting of

the pellets. Proper design of the transfer lines is also critical in terms of utilizing the optimum bend radii, blind tees, and proper angles. Consult your LyondellBasell technical service engineer for guidance in this area.

How to solve material handling problems

Since smooth piping is a leading contributor to angel hair and streamers, one solution is to roughen the interior wall of the piping. This causes the pellets to tumble instead of sliding along the pipe, minimizing streamer formation. However, as the rapidly moving polyolefin pellets contact an extremely rough surface, small particles may be broken off the pellets creating fines or dust.

Two pipe finishes, in particular, have proven to be effective in minimizing buildup and giving the longest life in transfer systems. One is a sand-blasted finish of 600 to 700 RMS roughness. This finish is probably the easiest to obtain. However, due to its sharp edges, it will initially create dust and fines until the edges become rounded.

The other finish is achieved with shot blasting using a #55 shot with 55-60 Rockwell hardness to produce a 900 RMS roughness. Variations of this finish are commonly known as “hammer-finished” surfaces. The shot blasting allows deeper penetration and increases hardness, which in turn leads to longer surface life.

The rounded edges obtained minimize the initial problems encountered with dust and fines. They also reduce metal contamination possibly associated with the sandblasted finish.

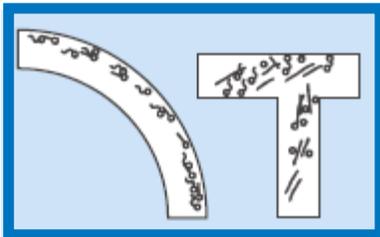
Whenever a new transfer system is installed or when a portion of an existing system is replaced, the interior surfaces should be treated by either

sand or shot blasting. The initial cost of having this done is far outweighed by the prevention of future problems.

Elimination of long-radius bends where possible is also important as they are probably the leading contributor to streamer formation. When this type of bend is used, it is critical that the interior surface should be either sand- or shot-blasted.

The use of self-cleaning, stainless steel “tees” in place of long bends prevents the formation of streamers along the curvature of the bend, causing the resin to tumble instead of slide (Figure 13). However, there is a loss of efficiency within the transfer system when this method is used. Precautions should be taken to ensure that sufficient blower capacity is available to prevent clogging of the transfer lines and maintain the required transfer rate.

Figure 13. Eliminate long-radius bends where possible. The use of stainless steel “tees” prevents the formation of streamers along the curvature of the bend.



To extend the life of the transfer piping, it should be rotated 90° at periodic intervals. Resin pellets tend to wear grooves in the bottom of the piping as they are transferred which not only contributes to fines and streamer formation but also accelerated wear due to non-uniform abrasion.

Regardless of the type of equipment used or the materials transferred, a transfer system should be maintained and kept clean in the same manner as any other piece of production equipment. Periodic washing and drying of silos and holding bins reduces the problem of fines and dust build-up due to static charges.

Other steps to eliminate contamination include:

- Inspect the entire transfer system on a regular basis
- Clean all filters in the transfer system periodically
- Ensure that the suction line is not lying on the ground during storage or when the system is started to prevent debris from entering the system
- Place air filters over hopper car hatches and bottom valves during unloading to prevent debris or moisture from contaminating the material
- Purge the lines with air and then with a small amount of product prior to filling storage silos or bins
- Allow blowers to run for several minutes after unloading to clear the lines and reduce the chance of cross-contamination of product.

Information regarding transfer systems and types of interior finishes available can be obtained from most suppliers of materials handling equipment or by consulting your LyondellBasell technical service engineer. Complete systems can be supplied which, when properly maintained, efficiently convey contamination-free product.

Other material handling practices

Beside-the-press vacuum loaders are used to feed many injection molding machines. These units draw resin pellets from drums or cartons placed beside the machine. In some set-ups, the vacuum loaders draw from multiple sources and directly feed the hopper with resin, regrind, colorants and other concentrate additives. Good housekeeping procedures are particularly important when working with beside-the-press loaders since contaminants can easily get into the material containers.

Blending with colorants, additives and other materials is done using on-the-machine blending units consisting of multiple hoppers feeding different resin compound ingredients. Colorants, additives, regrind and base resin are combined using either volumetric or, the more accurate, weight-loss feeding (gravimetric) techniques. Microprocessor controls monitor and control the amount of material fed into a mixing chamber below the hoppers. Recipe data can be stored in the control unit for instant retrieval.

Central blending units can also be used especially when much higher overall volumes are required. A central vacuum loading system transfers the finished blend to the individual molding machines.

The injection molding process

The injection molding process begins with the gravity feeding of polyolefin pellets from a hopper into the plasticating/injection unit of the molding machine. Heat and pressure are applied to the polyolefin resin, causing it to melt and flow. The melt is injected under high pressure into the mold.

Pressure is maintained on the material in the cavity until it cools and solidifies. When the part temperatures have been reduced sufficiently below the material's distortion temperature, the mold opens and the part is ejected.

The complete process is called a molding cycle. The period between the start of the injection of the melt into the mold cavity and the opening of the mold is called the clamp close time. The total injection cycle time consists of the clamp close time plus the time required to open the mold, eject the part, and close the mold again.

There are four basic components to an injection molding machine:

1. Injection unit/plasticator
2. Clamp unit
3. Injection mold
4. Control system

Injection units

Plunger injection units (Figure 14) were the first types used for injection molding, but their use today is quite limited.

The reciprocating screw injection molder is the most common molding machine in use today for mold-ing polyolefins. The injection unit (Figure 15) mixes, plasticates and injects a thermoplastic melt into a closed mold. The reciprocating screw accomplishes this in the following manner:

1. The injection cycle starts with the screw in the forward position.
2. Initially, the screw begins to rotate in the heated barrel. Resin pellets are forced by this action to move forward through the channels of the screw.
3. As the pellets move forward, they are tumbled, mixed and gradually compressed together as the screw channels become shallower. The section of the screw nearest the hopper is called the feed section, in which no compression takes place.
4. As the pellets travel down the barrel, they are heated by friction and the heat conducted from the external electric heater bands. The friction is caused by the pellets sliding against themselves and the inner wall of the barrel and the screw surface. The heat from the friction and conduction cause the pellets to melt. The majority of the melting occurs in the transition section of the screw, where compression of the polymer is taking place as the root diameter of the screw is increased.
5. Next, the melted polymer is further mixed and homogenized in the metering section of the screw. In the metering section of the screw, the root diameter has reached its maximum, and no further compression takes place.
6. The polymer melt flows in front of the screw tip and the pressure produced by the build-up of polymer in front of the screw causes the screw to be pushed backward in the barrel as it continues to rotate.
7. The screw stops turning when the volume of melt produced ahead of the screw tip is

Figure 14. Schematic cross-section of a typical plunger (or ram or piston) injection molding system

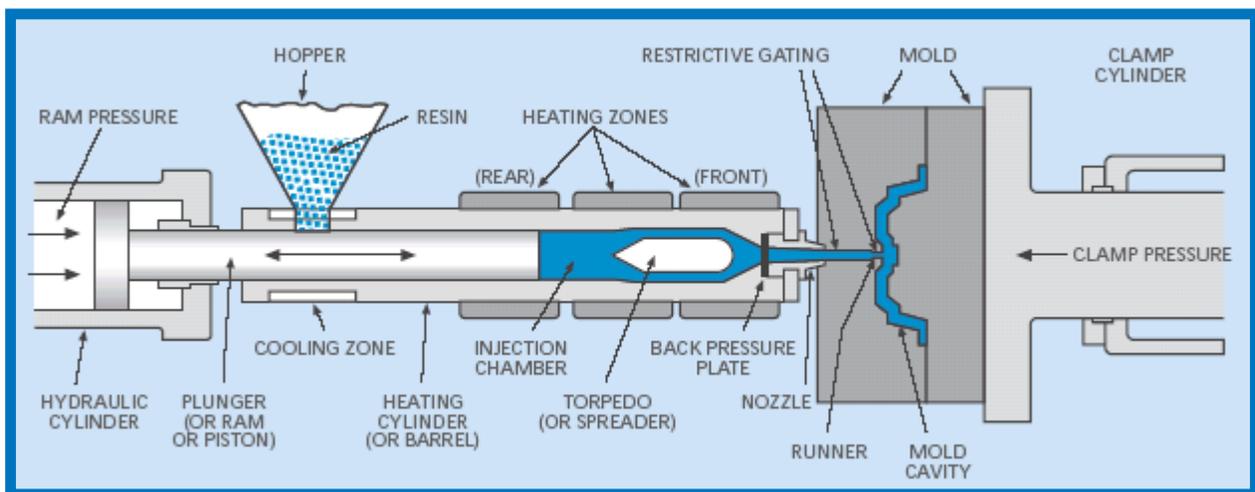


Figure 15. Schematic cross-section of a typical screw injection molding machine, showing the screw in the retracted (A) and forward (B) position

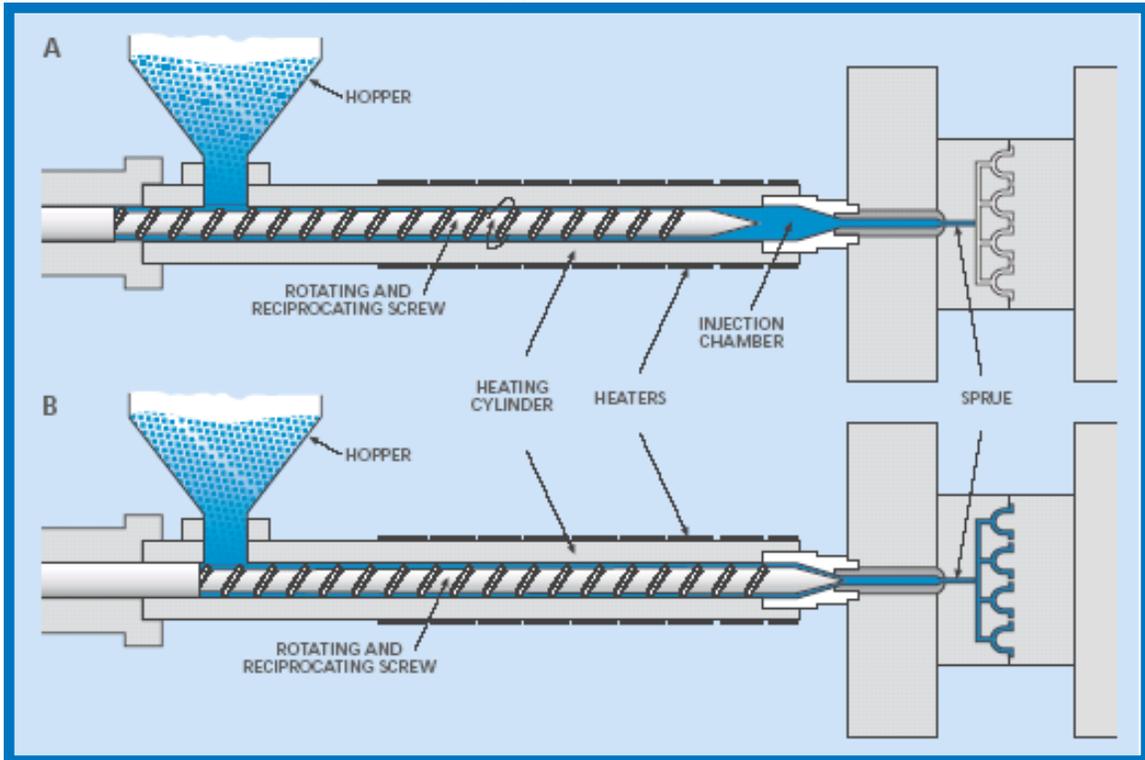
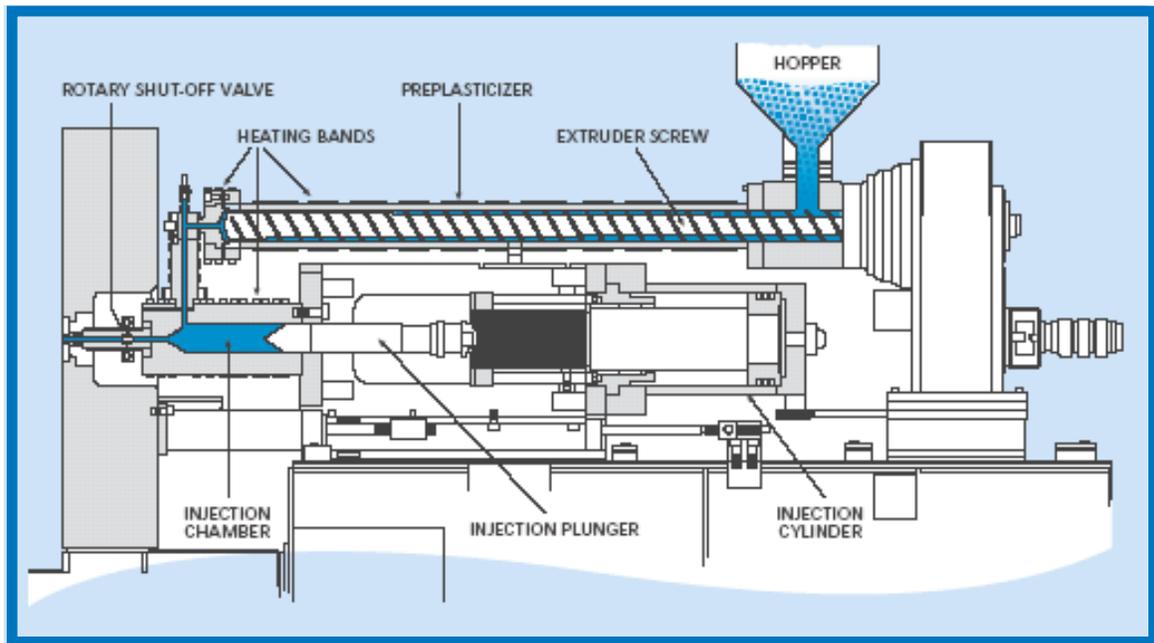


Figure 16. In this 2-stage injection molding machine, the screw-type preplasticizer is atop and parallel to the horizontal plunger injection cylinder and chamber



sufficient to completely fill the mold cavity and runner system (the channels leading to the mold cavity). This amount of material is called the shot size and the period during which the screw rotates is called the screw recovery time.

8. The screw is then forced forward, injecting the melt into the mold. This is called the injection stage.

In order to compensate for material shrinkage in the cavity due to cooling, an excess amount of material is generally held in front of the screw at the end of the injection stroke. This extra material is called the cushion and, during the packing phase, some of the cushion material continues to be slowly injected into the cavity to compensate for the volume lost due to the shrinkage of the material in the mold and the compressibility of the plastic.

Backpressure is the amount of hydraulic pressure applied to the back of the screw as it rotates. Varying the amount of backpressure alters the pressure exerted on the polymer in front of the screw. Increasing backpressure also changes the amount of internal energy transmitted to the melt by the shearing action of the rotating screw. An increase in backpressure raises the melt temperature without requiring an increase in heating cylinder temperatures and improves mixing and plasticating. Unfortunately, increasing backpressure also reduces screw recovery rates and can add unnecessary shear (heat) to the polymer which may lead to polymer degradation. Typically, backpressure is set at a minimum unless additional mixing is required.

Two-stage systems, also called screw preplasticators, are available

(Figure 16) in which the plasticating unit feeds a separate injection cylinder called an accumulator. Melt is injected into the mold using a ram in the accumulator. Machines equipped with accumulators can be used for molding parts requiring very large shot sizes, for the high-speed injection needed to fill long and narrow mold cavities, and for molding parts requiring better control of shot size and injection pressure.

Plasticator specifications

Injection capacity is defined as the maximum shot size in ounces (oz.) of general-purpose polystyrene (PS). In equating this to polyolefins, use approximately 90% – 95% of the capacity stated for PS. The plasticating rate is usually given in pounds/hour or ounces/second for PS. Because of differences in melting characteristics and different sensitivities to screw design variables, it is not possible to easily convert or apply

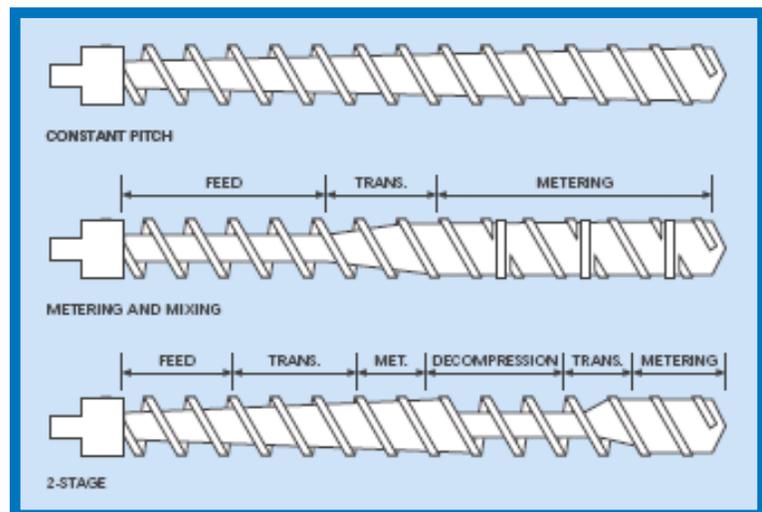
this value to polyolefins. Injection rate is the maximum rate at which the plasticized material can be injected through the nozzle in cubic inches/minute at a stated pressure.

Injection pressure is generally expressed as the hydraulic pressure in psi (pounds/square inch) applied to the screw during injection. The maximum injection pressure available varies and the actual pressure required depends on the resin, melt temperature, mold cooling, part design and mold design. Most plasticating units have a chart which relates the hydraulic pressure to the pressure actually applied to the polymer.

Screw designs

Numerous plasticating screw designs are available for injection molding polyolefins (Figure 17). However, since it is impossible to have a screw designed for every molding job, general-purpose screws are most commonly used. The shallower the screw channels, the smaller the resin volume conveyed to the tip of the screw.

Figure 17. Screw type configurations used in injection molding machines



On the other hand, while deep screw channels accommodate larger shot sizes more quickly, they do not heat and plasticate the melt as efficiently as a screw with shallower channels. The three basic screw sections are described in Table 3.

There are a number of barrier screw designs available which offer some benefits not provided by general-purpose screws. Barrier screws provide more efficient mixing without increased backpressure and, in some cases, recovery times may be decreased. These advantages are offset by the increased risk of black speck formation. The deep flights in a barrier screw may have stagnant areas in which there is a reduction in the flow of the material. The molten plastic tends to stay in these areas and degrade, ultimately causing black specks in the parts as the degraded material flakes off the screw. When purchasing barrier screws, it is recommended that the molder work closely with the screw designer to ensure that stagnant areas are avoided and that the screw is properly designed for the material being used.

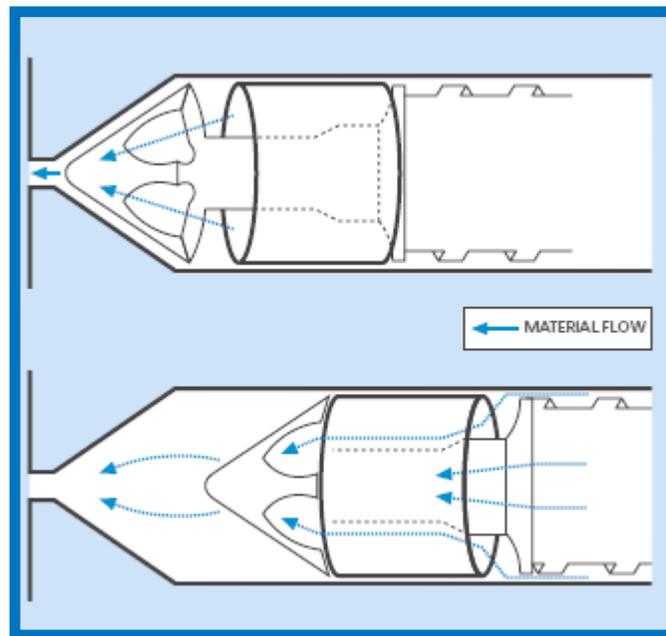
Plasticating screws for thermoplastics generally have interchangeable tips. The two most commonly used tips in the injection molding of polyolefins are sliding check ring and ball-check non-return valves. In the molding cycle, as the screw moves forward to inject material into the mold, the non-return valves close to prevent material from flowing back over the flights of the screw. Typical sliding check ring and ball check valves are shown in Figures 18 and 19.

Because of their tendency to wear, it is critical to periodically inspect the condition of sliding ring shut-off tips. Excessive wear will result in inconsistencies in shot size and melt temperature.

Table 3. Functions of the three sections of an extrusion screw

SECTION	CHANNEL DEPTH	FUNCTIONS
Feed	Deep (Constant)	Cool resin pellets are moved forward into hotter barrel zones and begin melting.
Transition	Decreasing (Tapered)	Resin is compressed, melted and mixed. Air carried along slips back to the feed section to be vented out the hopper.
Metering	Shallow (Constant)	Sufficient back pressure is created to make the melt homogenous (uniform), make its temperature uniform and meter it into the injection chamber.

Figure 18. Typical sliding check ring showing injection stage (top) and retraction stage (bottom)



The typical length-to-diameter (L/D) ratio for polyolefin reciprocating screws is about 20-30:1, with a compression ratio of 2-3:1. Longer screw lengths are generally preferred as they provide better homogeneity of temperature and melt quality.

Nozzles

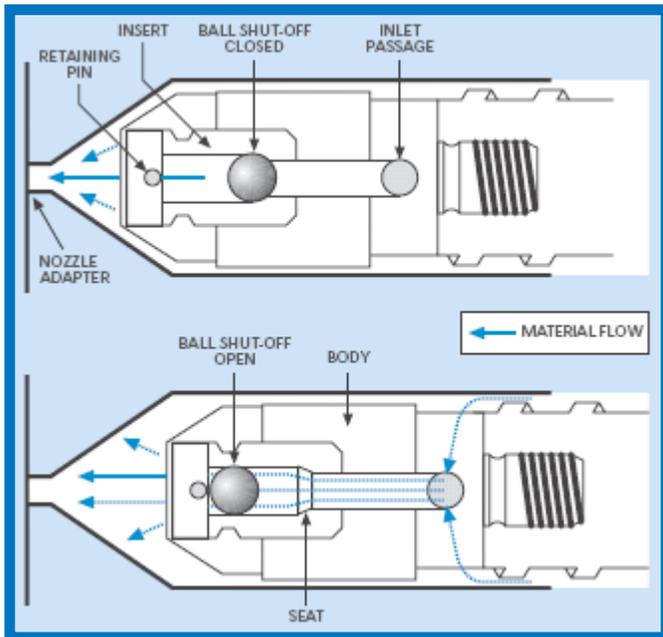
The injection-unit nozzle is connected to the barrel and directs the flow of the melt into the mold.

The nozzle extends into the fixed platen and mates to an indentation in the front of the mold called the sprue bushing.

The nozzle may have a positive shut-off device or it may be open and rely on the freezing-off of the melt in the gate areas of the mold to keep the resin from flowing back into the injection unit. Some nozzles may be connected to a temperature control device to control the melt temperature.

Clamp mechanisms

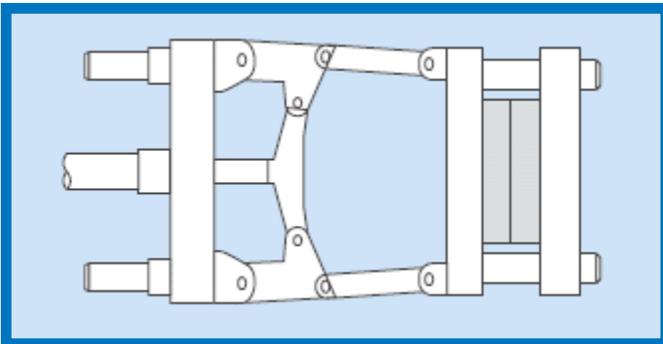
Figure 19. Typical ball check assembly showing injection stage (top) and front discharge-retraction stage (bottom)



There are three basic types of injection molding machine clamps: mechanical, also called toggle units, hydraulic and a combination of these called hydromechanical clamps.

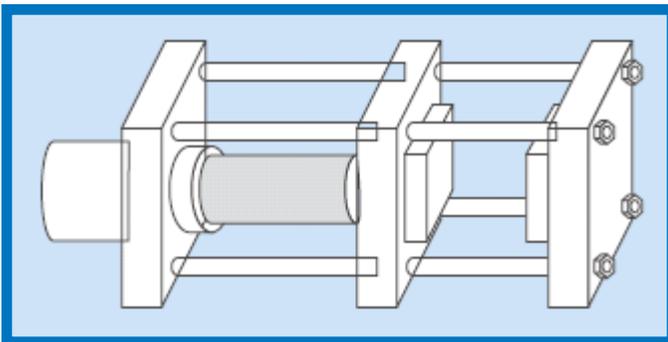
Toggle clamps, which are less expensive to build, are most widely used on small tonnage machines (typically, less than 500 tons). The toggle action can best be understood by looking at your arm when it is bent at the elbow and then when it is fully extended. In the toggle clamp, a hydraulic cylinder moves the unit's crosshead forward, extending the toggle links and pushing the platen forward. The mechanical advantage is low as the clamp opens or closes, which permits rapid clamp movement. This action slows and the mechanical advantage increases as the platen reaches the mold-close position. The slow speed is important for mold protection.

Figure 20. Toggle clamping system



Full clamp pressure is reached when the linkage is fully extended. To adjust the toggle clamp to different mold heights, the entire toggle mechanism and moving platen assembly are moved along tie rods. The position of the toggle mechanism depends on where the mold closes when the toggle is at full extension. The toggle opens when hydraulic pressure is applied to the opposite side of the clamp cylinder. See Figure 20.

Figure 21. Hydraulic clamping system



Hydraulic clamps generally are used on injection molding machines in the 150 ton to 1,000+ ton clamp tonnage range. In this type of clamp, hydraulic oil is used to move the platen through the full closing and opening strokes. The fluid is metered into a reservoir behind the main ram. At first quite rapid, the oil flow is slowed as the ram reaches the mold-close position in order to protect the

mold. An oil fill valve closes when the mold is closed. The area behind the ram is then pressurized to build full clamp tonnage. To open the mold, the oil valve is first partially opened to smoothly open the mold. Once the mold halves are separated, the clamp accelerates to a fast open speed (Figure 21).

Mold set-up is much easier with a hydraulic clamp than with a toggle clamp since hydraulic clamp tonnage can be reached anywhere along the clamp stroke. Mold set-up is accomplished by setting the clamp position from the machine's control center.

Hydromechanical clamps are commonly used on very large injection molding machines, i.e., over 1000 tons. In the hydromechanical clamp, a hydraulically actuated toggle mechanism pushes the moving platen at high speed to a point where the mold halves are nearly closed. A mechanical locking plate or links prevent rearward movement during final build-up to full clamp tonnage. Short-stroke hydraulic cylinders are used to move the platen the final short closing distance and develop full clamp tonnage. See Figure 22.

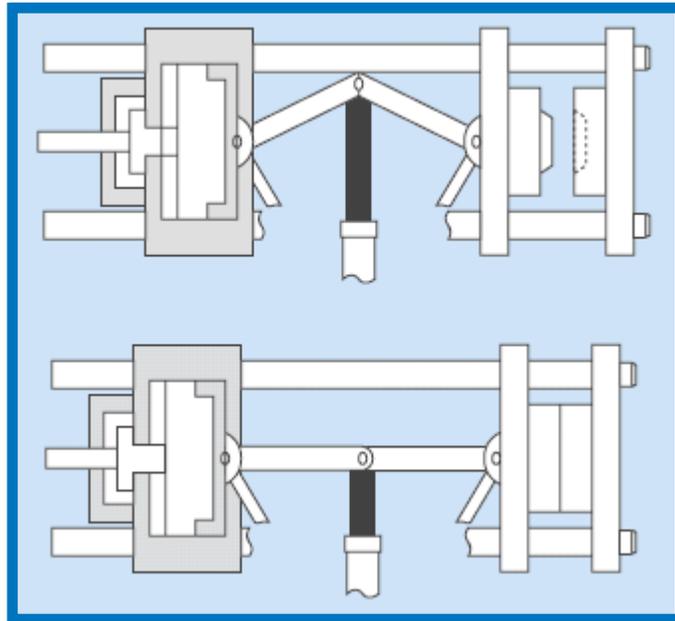
Clamp specifications

Key clamp specifications to consider in choosing an injection molding machine are:

- Clamp stroke
- Minimum mold thickness
- Clearance between tie bars
- Maximum daylight opening
- Platen size
- Clamp tonnage

Clamp stroke is the maximum distance (inches) the moving platen can travel. Clamp stroke is a major factor in determining the minimum mold thickness that can be used with the machine.

Figure 22 . Hydromechanical clamping system. Top view shows clamp open position of piston and ram. Bottom view shows clamp closed, toggled and ram pressurized.



Generally, clamp specifications also state the minimum mold thickness for which the clamp can develop its full tonnage.

Maximum daylight opening is the distance (inches) between the two platens when the clamp is completely open. This measurement is a major factor in determining the effective maximum mold thickness which takes into account the mold opening required for part ejection or removal. Complicated molds may require more opening space and rigid mounting surfaces, since high platen deflection under load could damage the mold. An allowable deflection of 0.001 in/ft of span with full clamp load on the center of the platen is, generally, considered acceptable.

Platen size is given in horizontal and vertical measurements (inches) for the full platen. Since there are tie-bars running through the corners of the platens, mold-size limits are less than full platen size. A mold can extend between the tie-bars in either the vertical or

horizontal direction but, generally, should not extend outside of the platens.

Clearance between tie-bars is given for the distance (inches) between the top tie-bars (horizontal) and sidebars (vertical). Since the tie-bars are fixed on most injection molding clamps, the distance between them dictates the maximum size of a mold that can be placed in the clamp.

Clamp tonnage is the maximum force which the clamp can develop. A clamping pressure of five-tons-per-square-inch of the projected area of the molding (including the runner system) is more than adequate for polyolefins. However, where packing is not a major factor, this pressure may be as low as 2 tons/in². An industry rule-of-thumb is that a clamp force of 2 to 3 tons per in² of the projected area of the part(s) and cold runner system is adequate for reciprocating-screw-

type, injection-molding machines. Some thin-wall stack molds may require 5 tons/in² for optimum performance.

Injection molds

There are many types of injection molds and tooling in use today, such as two-plate, three-plate and stack molds. Two and three-plate molds are more commonly used for heavy wall and non-packaging products. Both cold and hot-runner systems are used for two and three-plate molds. All stack molds use a hot manifold to convey the melt to the cavities. Each mold component must be machined and finished to exact dimensions with very tight tolerances and must be heat-treated to be able to withstand very high injection and clamp pressures. Injection molds are the most expensive molds used in plastics processing with very long lead times required for their design and fabrication. Every mold must be tested and debugged to prove-out the ejection system, cooling and/or heating system and operating components before it is placed in production.

Types of molds

A two-plate mold (Figure 23) has only one parting line. If a runner is used, it is connected to the molded product and requires manual removal and separation after the part is ejected. This type of mold is the least expensive to make and is commonly used for parts with relatively simple geometries.

Three-plate molds (Figure 24) have two parting lines, one for the runner system and one for the molded product. When the mold opens, the runner is automatically separated from the product to allow separate handling. This eliminates the need for manual separation and removal and the sprue and runner system may be

Figure 23. Two-plate mold

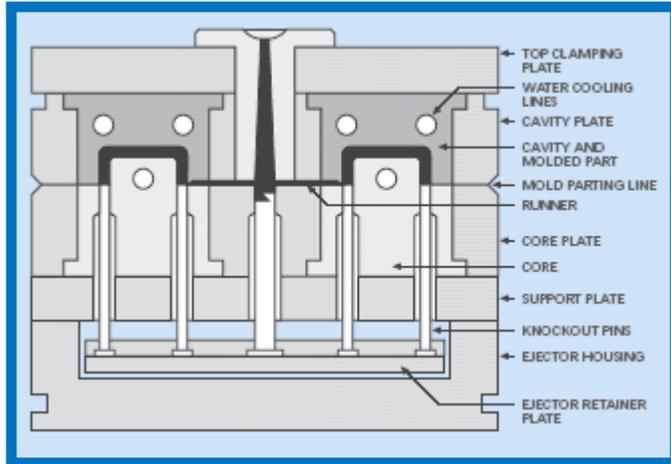
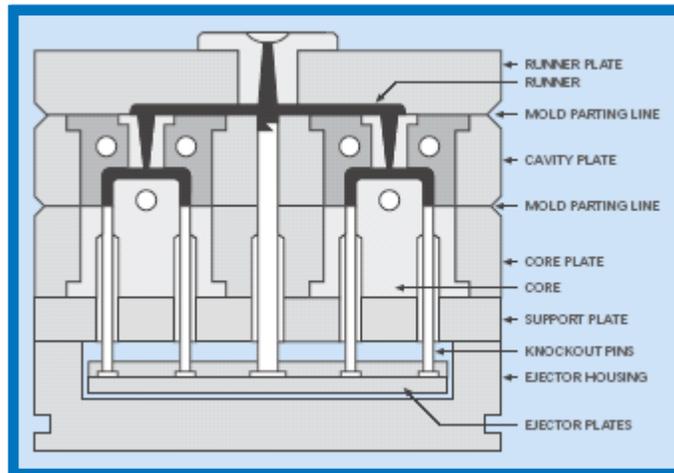


Figure 24. Three-plate mold



fed directly to a recycling system. This type of mold is more expensive than the two-plate mold.

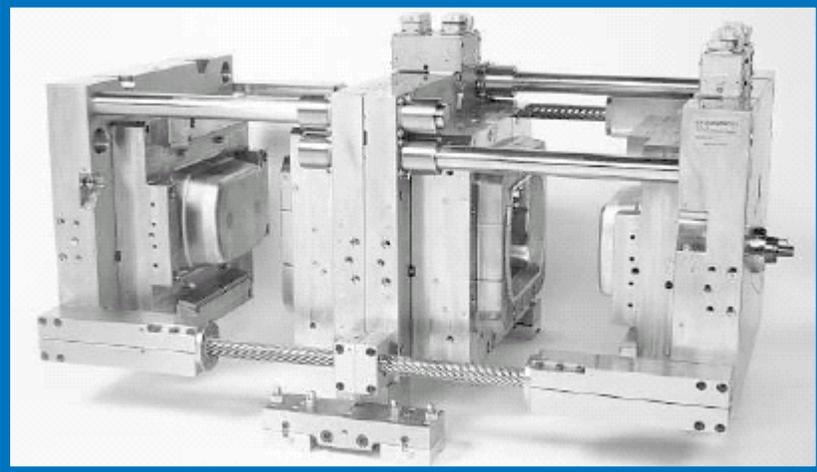
Stack molds (Figures 25 and 26) can be two, three or four levels. The advantage of the stack mold is that it can, generally, produce a larger number of products versus a two or three-plate mold utilizing the same machine clamp tonnage. The disadvantage is that it requires a molding press with much greater daylight opening to accommodate the mold height. This type of mold

is much more expensive and takes longer to build. Three level stack molds are very new and four level stack molds have been around for less than five years. The dairy container and lid industries commonly use stack molds. The four-level is common for lids, and the two-level is common for containers.

Sprues and runners

The sprue and runner system is the conduit that connects the machine nozzle to the cavities.

Figure 25. 2 x 1 wash basin stack mold *Photo courtesy of Tradesco Mold, Ltd.*



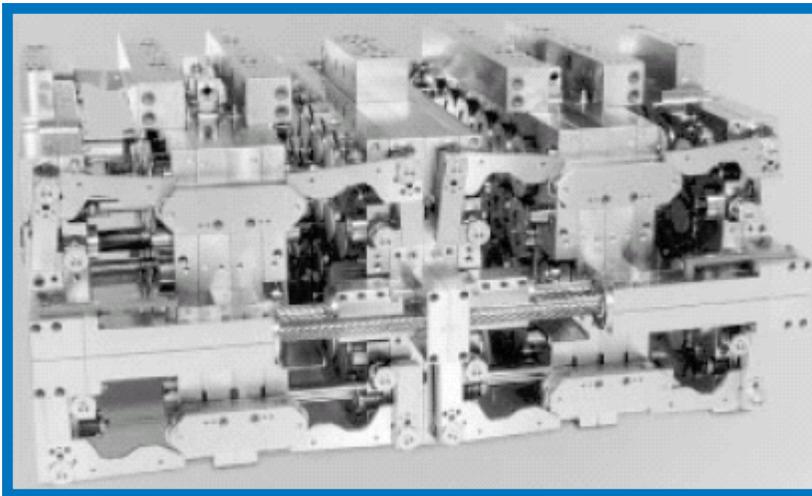
During the injection phase of the molding cycle, the molten material flows through the sprue and runner to the cavities.

The sprue connects the machine nozzle to the runner and may be either a cold or a hot sprue. In the hot sprue design, the sprue has heating elements that maintain the plastic in a molten state eliminating the need for separation and reclamation. Ideally, the sprue should be as short as possible to minimize the pressure loss during injection. A cold sprue is tapered for easy release from the mold.

There are three basic runner types in use:

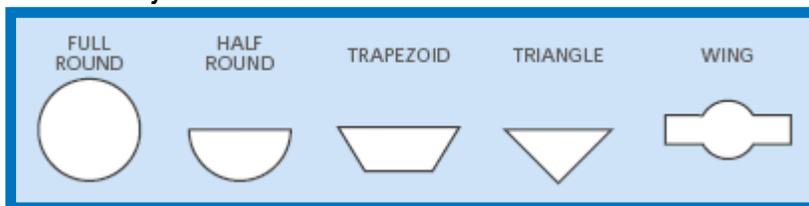
- Cold Runner
- Insulated Runner
- Hot Runner

Figure 26. 4 x 24 stack mold *Photo courtesy of Tradesco Mold, Ltd.*



Cold runners are commonly used in two and three-plate molds, but not in stack molds which require the use of a hot runner. The most commonly used runner designs are full-round, half-round and trapezoidal (Figure 27). The full-round is generally preferred for ease of machining and lower pressure loss. For fast cycles a full-round is not recommended since the greater mass of material takes longer to cool and may control the cycle time. The runner should be polished for ease of mold filling and part ejection.

Figure 27. Schematic showing typical runner designs found in injection molds



The insulated runner (Figure 29) is the precursor to the hot runner. The runner diameter is very large and a thick skin is formed on the outside of the runner. The molten plastic flows in the center and, due to external insulation and the low thermal conductivity of the polymer, remains molten during the cycle. This design eliminates the need for removing and/or recycling the runner. The problem with this design is that when the machine is down for any extended period of time the runner solidifies and has to be physically removed before

Figure 28. Insulated runner system

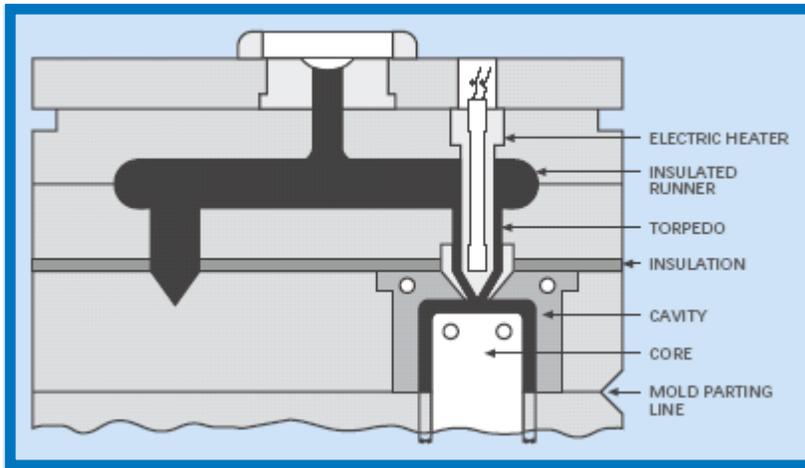
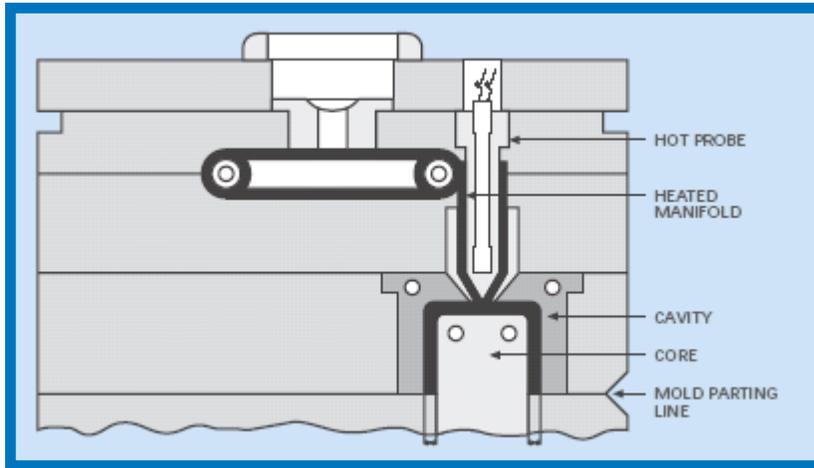


Figure 29. Hot-runner system



beginning the next molding cycle. As molders have become more comfortable with hot runner technology, insulated runners are rapidly becoming obsolete and not many molds are built today utilizing this technology.

The externally heated hot-runner system (Figure 29) also maintains the plastic in a fluid state while the mold is running with the pressure at each gate approximately the same. Maintaining a uniform temperature in the sprue bar and the hot-runner manifold is very critical to process and product consistency. Start-up procedures must be carefully followed

according to the mold maker's specifications to prevent damage and material leakage in the manifold.

Mold venting

When molten plastic is injected into the mold, the air in the cavity has to be displaced. To accomplish this, vents are machined into the parting line to evacuate the air and are extremely important to the consistent production of high quality products. In many cases, this is an area in mold design and construction that is often overlooked.

Vents should be located at the extremities of the part and at locations where melt flow fronts come together. Venting is also easily achieved around ejector pins and core slides provided that there is sufficient clearance between the pin/slide and the mold. Typical mold vents are channels cut from the cavity or runner straight to the edge of the mold. Closest to the part, they are typically 0.0005-0.001 in. (0.013-0.025 mm) in depth and 0.063-0.5 in. (1.6-12.7 mm) in width. The initial vent thickness should be maintained for about 0.5 in. (12.7 mm) and then the depth can be increased to about 0.003 in. (0.076 mm) to the edge of the mold. The vents should be polished towards the edge of the mold to make them 'self-cleaning.' Build-up in the vents will eventually affect mold filling resulting in non-uniform fill and unbalanced cavities. For this reason, it is important that vents be inspected between production runs to ensure that they are clean and within specification. In some cases, reduction of the injection rate prior to final filling of the cavity will prevent burning and also prevent the mold from opening.

Continuous parting-line venting may be necessary in high speed molding operations. Even though burning is not evident, the lack of burn marks does not ensure that the molds are properly vented. Increasing vent areas may help reduce cycle time. Proper venting will also aid mold filling by decreasing the resistance due to air pressure on the flow front.

Mild sand blasting or vapor blasting of the mold cavity assists in venting and part release. However, for high-gloss finishes, this blasting is not advisable. Vapor honing may help alleviate a venting problem area but care must be taken that honing is not too deep or wide to be noticeable on the finished part.

Gating

The gate is the bridge between the runner and the cavity. Depending on the specific material and part design (wall thickness, geometry, etc.) there are many different types of gates which can be used (Figure 30).

The type and size of gate are very critical since they can affect many factors including mold-fill time, overall cycle, orientation, shrinkage, warpage, and part appearance.

Because it acts as a restriction to the polymer flow, a high shear rate is created at the gate often resulting in a temperature increase. There is also a high pressure drop across the gate which needs to be overcome by increased injection pressure or higher temperatures. The pressure drop can be reduced, to a certain degree, by using shorter gate land lengths.

A large gate provides easy filling with relatively low shear rates and pressure drops. However, if it is too large, it will require an excessive amount of time to cool, lengthening the cycle. It is also possible that insufficient packing and subsequent sinks or voids will occur if either sections of the part or the sprue and runner system freeze off before the gate.

A gate which is too small will require higher pressures to inject the material and may cause problems in part filling. If the gate freezes off before the part cools, it will not be possible to develop adequate packing, resulting in voids or sinks. With extremely small gates, jetting or melt fracture of the polymer flow will cause surface appearance defects, including delamination.

To ensure uniform fill, it is critical that the feed system (sprue,

runners, and gates) be balanced. This depends on the size and location of the gates and is often determined by experience. Advances in mold filling simulation software have provided an additional tool for analysis prior to the manufacture of the tool. Fine-tuning may be required and is generally done by utilizing a series of short shots, observing the fill pattern, and making minor adjustments, as required. For multi-cavity tools utilizing single gates and a hot runner system, adjustment of the temperatures of the individual gates may be used to balance the overall fill pattern.

In high-speed, thin-wall molding, it is common to provide cooling around the gate to remove the heat produced by the high shear rates. This may be supplemented by the use of inserts fabricated of high conductivity alloys, such as beryllium-copper, in these critical areas.

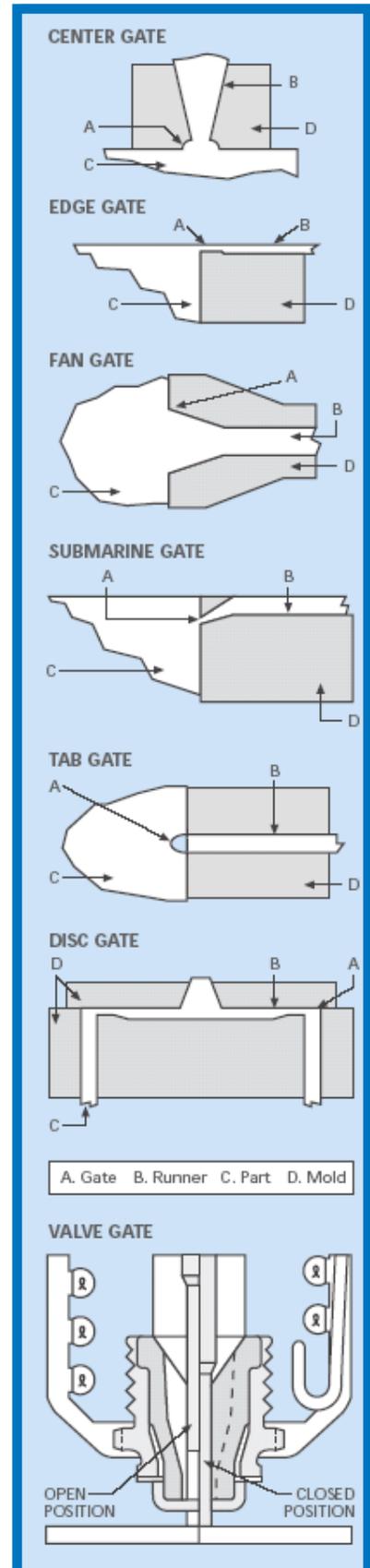
Mold cooling

Although mold cooling is extremely critical to cycle time, warpage, molded-in stresses, mold-filling, etc., the sizing and layout of the cooling pattern are often over-looked and neglected aspects in the initial stages of tool design.

The cooling layout should be considered in relationship to the thickness profile of the part and the general filling pattern in order to provide adequate cooling in critical sections and not overcool others which may cause part warpage. In areas where coolant flow may be restricted due to part geometry i.e., bosses, the use of inserts fabricated from high thermal conductivity alloys, such as beryllium-copper, should be considered.

In all cases, cooling channels should be sized in relation to the available coolant flow to ensure

Figure 30. Gating systems



turbulent flow which is much more effective for heat removal than lowering the temperature of the coolant. Routine inspection and acid-cleaning of cooling channels are recommended to maintain the coolant flow velocity and minimize pressure drops. Ideally, the temperature differential between coolant inlets and outlets should be about 2°F. Jumpers between cooling circuits should be avoided in order to reduce temperature differentials in the coolant.

The utilization of low pressure-drop manifolds, valves, fittings, etc. and in-line flowmeters and temperature indicators are also good practices to provide information regarding the efficiency and condition of the cooling system.

Ejection devices

The ejection of injection molded parts is most commonly accomplished by air, vacuum, pins or strip-per plates. Depending on part design, combinations of these systems are used for rapid positive ejection. Care should be taken in selecting ejection surfaces because of aesthetic and moldability requirements. Wherever possible, the part should be ejected off the core. For small, thin-walled moldings that may shrink onto the core, air ejection through the core is usually adequate for part removal. On some products with threaded or undercut features, collapsible, retractable or unscrewing cores are used.

Spiral flow measurement

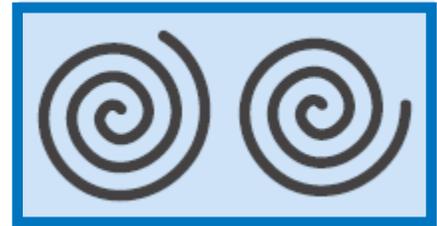
The relative processability of an injection molding resin is often determined by its Melt Index (MI) or Melt Flow Rate (MFR). This involves measuring the relative flow of the molten resin through a specified capillary in a calibrated laboratory instrument, while

maintaining the molten resin at 190°C (374°F) and 43.5 psi for Polyethylenes or at 230°C (446°F) for Polypropylenes.

Melt index is a good measurement of a resin's relative flow properties at low shear rates, but only for resins of the same molecular weight distribution (MWD). Under actual injection molding conditions, differences in MWD will affect the resin's melt viscosity (flow characteristics) at high shear rates. Temperatures, pressures and shear rates of actual molding do not conform to those of the MI or MFR test methods.

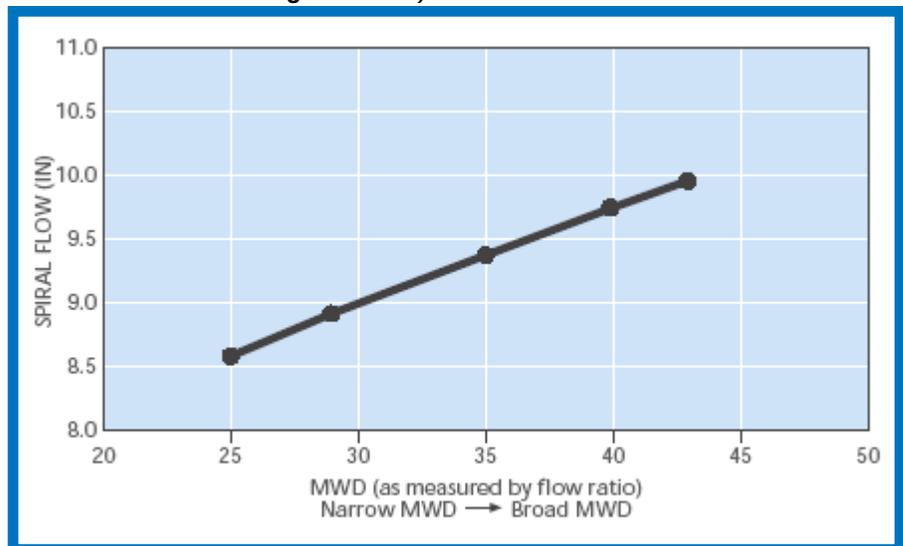
LyondellBasell has a number of unique manufacturing processes available which allow the control not only of the melt index and density, but also MWD. This capability results in a better overall balance in resin properties and processability. Because melt index and MWD play a key role in performance in actual end-use applications, LyondellBasell has utilized "Spiral Flow" (SF) as a more practical method of measuring and comparing a resin's performance using realistic processing conditions.

Figure 31. Broad MWD (left) and narrow MWD (right) spiral flow



Spiral flow measures the flow length when molten resin is injection molded into a long, 0.0625" radius, half-round spiral channel (Figure 31). The higher the spiral flow number (SFN), the easier the resin is to process. The melt temperature is monitored and maintained at 440°F (227°C) and injection molding is conducted using a constant pressure of 1,000 psi (7,000 kPa). Spiral flow is a more realistic measurement than melt index because it is run at a much higher shear rate allowing resins of similar MIs and different MWD to be compared at realistic conditions. The broader MWDs resins exhibit lower melt viscosity (higher SFN) at higher shear rates than narrower MWD resins with similar melt indices (Figure 32).

Figure 32. Effect of MWD on spiral flow of HDPE (all materials have MFR - 5 gms/10 min)



Since it does not take into account the effects of MWD, relying only on the melt index can be misleading.

For example, ALATHON[®] ETP H 5057, a broad MWD, 57 melt index resin for thin-wall HDPE applications, exhibits flow properties similar to many narrow MWD resins in the 75 to 80 melt index range.

LyondellBasell has established the use of spiral flow as a specification for all high-flow (30 melt index and above) HDPE resins and has begun reporting the spiral flow number for each lot on the Certificate of Analysis (COA). This allows the molder to compare the spiral flow of an incoming lot of resin with the SFN of the lot on-hand and readily estimate how the new lot will process relative to current production. For example, if the current lot being run has a SFN of 20 in. and the new lot has a reported SFN of 22 in., the new lot can be processed at either lower temperatures and/or at a faster production rate. Only minor adjustments in either melt temperature and/or injection pressure may be required to compensate for SFN variability from lot to lot.

General injection molding operating procedures

Prior to starting up the injection-molding machine, be sure to have the following available:

- Safety glasses for all personnel assisting in the start-up.
- Loose fitting, heavy-duty insulated work gloves.
- A large metal container or cardboard for collecting melt produced during the start-up procedure.
- Soft beryllium-copper, bronze, or aluminum tools for use in removing any plastic from the nozzle area.

Always refer to the manufacturer's operating manual for any specific start-up and shutdown procedures.

Refer to the LyondellBasell suggested resin startup conditions (Table 4) for general guidelines to use in starting up an injection molder on polyolefins.

General safety

As with any process involving energy and mechanical motion, injection molding can be a hazardous operation if appropriate safety procedures aren't well documented and followed. (Refer to the Manufacturer's operating manual.)

Mechanical, electrical, and hydraulic interlocks are critical to

the safe operation of any piece of processing equipment. In some cases, these interlocks may need to be bypassed while performing set-up and maintenance functions. Under no circumstance should this be done by non-qualified personnel. In order to assure utmost safety during normal operation, interlocks should never be bypassed.

Keep all molding equipment and the surrounding work areas clean and well maintained.

Hydraulic leaks should be repaired immediately to eliminate safety hazards. Hydraulic lines, valves, fittings and hoses should be checked periodically per the manufacturer's recommendations.

Table 4. Suggested start-up conditions (based on general purpose melt index/flow rate products)

PRODUCT	CYLINDER TEMPERATURE				
	EVA	LDPE	LLDPE	HDPE	PP
Rear	320°F	325°F	350°F	450°F	400°F
Center	340°F	340°F	375°F	470°F	425°F
Front	340°F	350°F	400°F	475°F	450°F
Nozzle	340°F	350°F	400°F	475°F	450°F
Higher temperature settings may be necessary for parts or cycles requiring more plasticizing capacity. EVA's with higher VA incorporated may require less heat.					
Mold Temperature	40-70°F	45-75°F	45-75°F	45-65°F	60-80°F
Mold temperature may be raised to improve flow and surface finish or lowered for faster cycles, lower shrink and better ejection.					
Mold Cycle					
Injection	15 sec.	15 sec.	15 sec.	15 sec.	15 sec.
Booster	0-5 sec.	0-5 sec.	0-5 sec.	0-5 sec.	0-5 sec.
Cure	15-30 sec.	15-30 sec.	15-30 sec.	15-30 sec.	15-30 sec.
Injection time and cure will vary with part thickness. Reduce injection time as much as possible. Follow by reducing cure time.					
Screw Rotation Speed should be adjusted to provide a 15-30 second retraction time.					
Screw Back Pressure – set slightly above minimum. Higher screw back pressure may be necessary for parts or cycles requiring more plasticizing capacity.					
Injection Speed – Set at maximum. May need to be slower for thicker parts or smaller gates.					
Injection Pressure – Minimum PSI without shortshot (adjust to fill cavity). Second stage may be lower for holding.					

Good housekeeping is essential. Loose pellets, tools, oil, etc. on and around the molding machine can cause accidents, damage to the equipment, or contamination of the parts.

Heat

High temperatures are necessary in the injection molding process. Always use heat-resistant gloves, safety glasses and protective clothing. Modern injection molding machines have warning signs identifying specific hot areas on molding machines; do not ignore these signs. Keep the splashguard in place during purging and when the machine is operating.

Polymer left in the barrel or hot runner system may often be pressurized. Care should be taken whenever resin flow is interrupted due to blockage or mechanical problems.

If the molding machine will be shut down for an extended period of time (30 minutes or longer), lower the heats, purge the machine or cycle it until the lower temperature is reached before shutting it down, leaving the screw full of resin and in the retracted position.

Electricity

Molding machines utilize high electrical voltage and have warning signs pointing out electrical hazards; do not ignore these signs. Keep water away from these areas. A periodic inspection of all electrical devices and connections for wear, looseness, etc. is very important.

Machinery motion

There is considerable mechanical motion during the injection molding process. Neckties and loose fitting clothing should not be worn around molding machines since these can be caught by the equipment movement and lead to physical

injury. Do not reach around, under, through, or over guards while the equipment is operating.

Be sure all people working near the injection molding machine know where the Emergency Shut Off but-on is located. Never disengage any of the safety mechanisms or inter-locks on the injection molding machine.

Some machines store energy (hydraulic, pneumatic, electrical, or gravitational) which can be present even when the machine is turned off. Consult the manufacturer's operating guide for methods to properly de-energize the equipment. As with any piece of potentially hazardous equipment, a suitable lock-out/tag-out procedure should be implemented and enforced.

The injection molding process and its effect on part performance

The molding cycle

As detailed in the section on the Injection Molding Process, there are several steps in the production of injection molded parts. In most cases, the injection molding cycle begins with the mold open, ejector pins/slides retracted, and the screw/ram ready with the next shot of material. The cycle then proceeds as follows:

1. **MOLD CLOSE:** Mold closes and clamp develops full closing pressure
2. **INJECTION:** Material is injected into the mold cavity
3. **PACKING:** Material is packed into the mold to fill out the part
4. **PLASTICATION:** Screw begins to rotate (or ram retracts) to develop the next shot of material

5. **COOLING:** Coinciding with the start of plastication, the cooling cycle begins (Note: Since coolant continuously circulates through the mold, cooling technically starts as soon as the melt contacts the cavity during injection)
6. **MOLD OPEN:** Mold opens, slides retract, ejector pins activate, part(s) are ejected.

The length of each of these steps will depend on the complexity of the mold, the size of the machine, and the geometry and end-use requirements of the part. A typical cycle for a four cavity, 16 oz. stadium cup can be found in (Figure 33) while one for a single cavity, 12 lb. bumper fascia can be found in (Figure 34).

Regardless of part size, weight, or mold complexity, nearly half of every injection cycle is spent cooling the part(s) to a temperature sufficient to allow ejection without post-mold distortion. Factors that affect the cooling rate of the part(s) will be covered later.

In almost all cases, part quality is the result of steps 2-5. The highest quality parts begin with a homogeneous plastic melt in terms of temperature and composition. Therefore, the quality of the next part to be produced is the result of the development of the shot during the current cycle. Shot size should be sufficient to produce a cushion of material at the end of the step 3 of 0.1-0.5 in. (2-13 mm). This cushion will keep the screw from 'bottoming out' and help maintain plastic pressure within the cavity.

Achieving a homogeneous melt is controlled by many factors including: screw design, screw speed, screw and barrel wear, back pressure, shot to barrel capacity ratio, soak time, and heater band settings.

Figure 33. Injection molding cycle for a 16.0z. stadium cup (4 cavity, HDPE, 31g each, 7.8 sec. cycle)

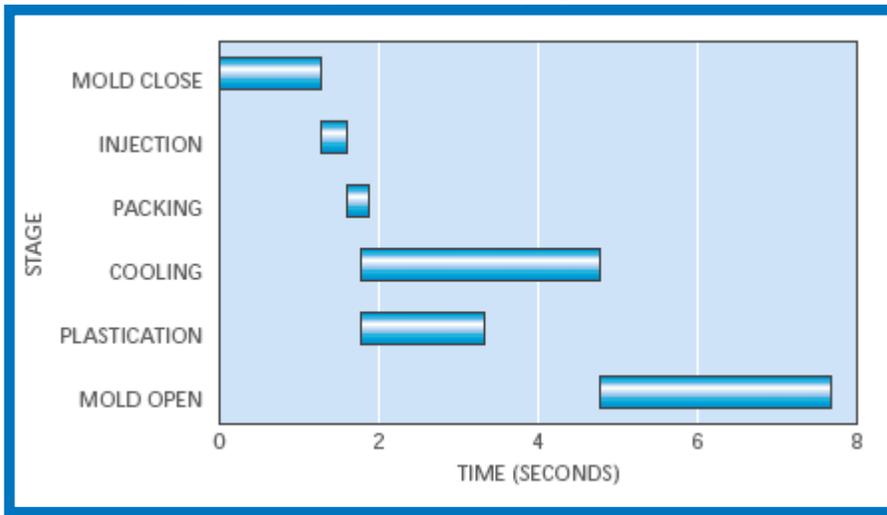
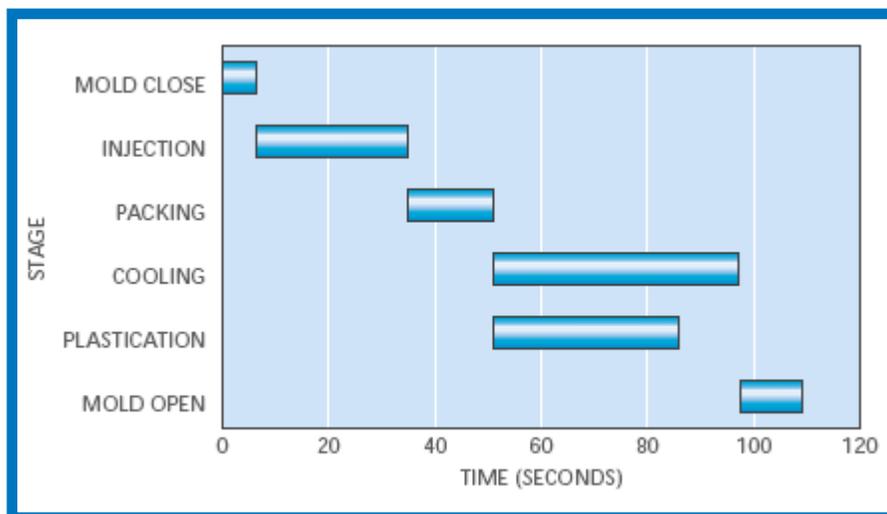


Figure 34. Injection molding cycle for automotive fascia (12 lb. shot, 0.125 in. thickness, 107 sec. cycle)



Screw design for polyolefin processing was covered in a previous section. A worn barrel and/or screw creates an increased gap between the two resulting in resin moving backward or staying in place instead of being conveyed forward by the screw. A worn screw and/or barrel can lead to poor melt consistency, degradation of the resin, and shot inconsistency. Screw speed

should be set so that the screw consistently recovers 1-2 seconds before the mold opens.

Backpressure is typically set at the minimum level that delivers a homogeneous melt (no unmelted pellets leaving the nozzle). However, backpressure may need to be increased to improve temperature consistency in the melt and to minimize or eliminate streaks due to poor dispersion of colorant. During plastification, most of the energy

provided for melting the resin pellets comes from shear heating due to friction between the pellets, screw and barrel. As the back pressure is increased, the screw works the material more in order to convey it forward, thereby raising the temperature of the melt more quickly. The increased work by the screw also increases the mixing of the molten plastic resulting in better temperature and homogeneity of the melt. However, too much back pressure can result in degradation of the plastic, an increase in the screw recovery time, increased energy costs, and more wear on the screw and barrel. Shear heating is also dependent on the viscosity of the plastic, screw design, screw speed, and back pressure. The latter two can be varied to some extent by the processor to control the shear heating and melt temperature.

The shot-to-barrel capacity ratio (SBCR) can also have an effect on melt, and therefore part quality. The ideal range for the shot to barrel capacity ratio is 30-60%. If the SBCR is less than 30%, too many shots of material reside in the barrel under the influence of heat from the heater bands and shear from the screw. This may lead to over-heating and degradation of the resin. If the SBCR is greater than 60%, less than two shots of material are in the barrel, which typically does not allow the melt temperature to equilibrate. A high SBCR will also mean that the screw may recover (develop the next shot) just before the mold opens which can lead to cycle alarms due to inadequate shot size. A SBCR range of 30-60% will provide adequate time for the melt temperature to equilibrate.

In order to achieve proper melt homogeneity, all of the pellets should be melted by the time they reach the middle of the transition zone on the screw. Figure 35

depicts the amount of energy needed to process a polypropylene impact copolymer. In order for the pellets to be fully melted halfway through the transition zone, 71% of the energy to reach the desired melt temperature (in this case, 450°F) must be transferred to the polymer.

The heater bands on the barrel provide only a small amount of the energy needed to melt the plastic. Most of the energy from the heater bands maintains the barrel temperature during processing and raises the temperature of the solid pellets in the feed zone. There are four typical temperature setting patterns for injection molding on a barrel with five heater zones (Figure 36):

1. Increasing: This pattern has the lowest temperature setting at the feed throat and the highest at the front of the screw with a steady increase of the temperature settings in between. The nozzle is typically set at the same temperature as the front zone. This pattern is the one most commonly used and is particularly recommended for lower melting point materials (such as EVA or EMA) to prevent bridging at the feed throat of the extruder. It is also recommended when the SBCR is low, typically <30%.

2. Decreasing: This pattern has the desired melt temperature at the front of the screw and the highest temperature at the back of the screw. The nozzle is usually set at the same temperature as the front zone. This temperature profile is recommended when the SBCR is >50% and screw recovery and residence times (time from resin entering the extruder to leaving the nozzle) are short. Sufficient feed throat cooling must be provided to prevent bridging. Otherwise, a low

Figure 35. Energy needed to bring PP to melt temperature

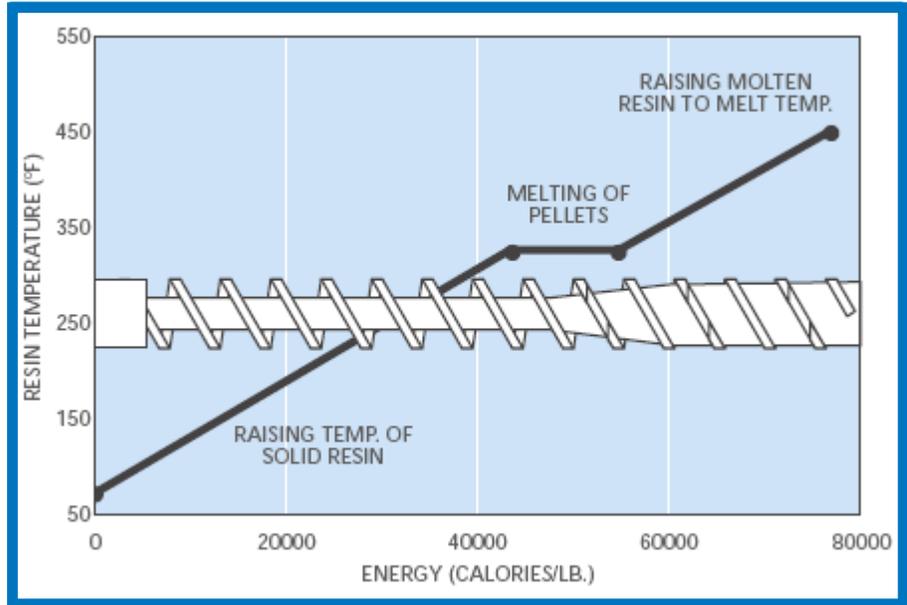
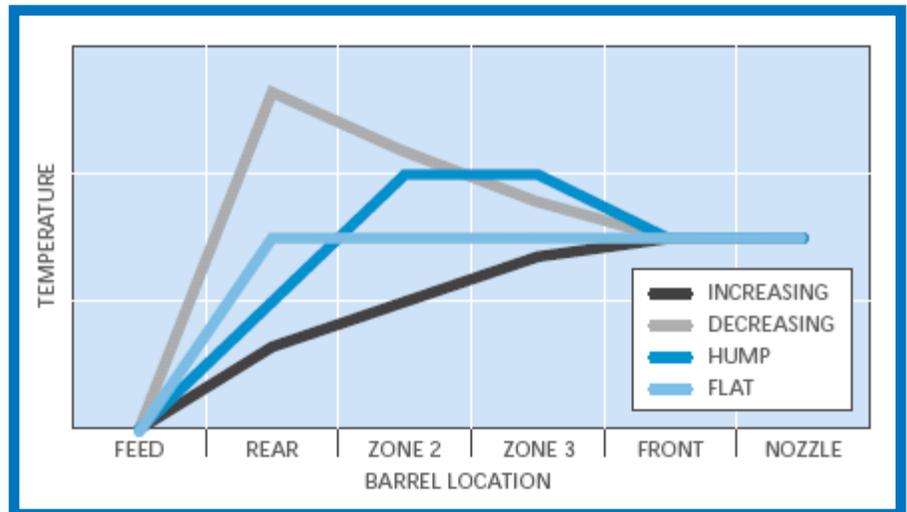


Figure 36. Various barrel temperature profiles



temperature set point should be used at the feed throat. In addition, this profile can increase the chance of air being entrapped in the melt instead of venting back through the hopper.

3. Hump: This pattern has the highest temperature settings (typically 20+ degrees higher

than desired melt temperature) in the middle of the screw to correspond with the transition section where the majority of the melting takes place. Settings near the feed throat are typically at the softening point of the resin while the settings at the front of the barrel and the nozzle should be at the desired melt

temperature. This profile is recommended when SBCR is 25-50% and overall residence time is 2 to 4 minutes.

4. Flat: This profile uses the desired melt temperature as the settings for all of the barrel zones except for the feed throat, which should be set at or below the softening point of the resin. This profile is typical of processes where the SBCR is 20 to 40%.

In actual practice, the specific screw design also plays an important part in obtaining the desired melt. It is possible that different screw designs may require different profiles to achieve similar melt characteristics even if they have the same or an equivalent SBCR.

By varying the screw speed and back pressure, shear heating is, for the most part, an easily controlled source of heat to the material. The best parts are typically produced when there is a balance of shear heating and heat from the heater bands. Once a temperature profile is chosen, it is recommended that the processor monitor current flow into the heater bands. The proper balance of shear heating and thermal heat is achieved when the current cycles regularly (typically several times a minute). This is of particular importance in the transition zone of the barrel; if the barrel is divided into quarters along the length, the transition zone is typically the middle two quarters of the barrel. Typically the heater bands in the feed zone will cycle regularly or be on nearly all the time. The heater bands in the metering zone and nozzle should cycle regularly but less frequently than the feed zones. Because accurate temperature control is so important, it is always a good practice to routinely check the calibration of the controllers.

If a heater band is on all the time, either the set point is too high to be reached or there is a problem with the thermocouple or heater band (it is working but not reading the actual temperature). If the thermocouple is fine and the desired setting for the zone is not out of line, the processor can increase the temperature settings upstream (closer to the feed throat) of the zone in question. Should this not reduce the time that the heater band is on, the temperature setting on the zone should be lowered until the band cycles regularly. This will prolong the life of the heater band and reduce the energy usage.

If a heater band does not draw current, or does so infrequently, there are two possible problems. Either most of the heat going into the plastic at this barrel zone is via uncontrolled shear heating or the thermocouple is broken, both of which should be corrected. A broken thermocouple will typically read out the maximum permissible temperature. If all of the cavities are filling with acceptable cycle times, the heater band set points in the zones upstream of the zone in question should be reduced. If this fails to get the heater band cycling, reset the upstream zone(s) to their original temperature(s) and increase the temperature set point.

It may appear that the procedures above are only serving to increase the overall melt temperature. While this is true to a small extent, the benefit is that they aid in providing a more homogeneous and controllable melt temperature that will improve the molded parts.

Now that we have a homogeneous melt stream in the barrel or accumulator, we need to examine the introduction of the plastic into the mold. The viscosity, or resistance to flow, of the resin is affected by temperature and shear rate. Increasing the melt temperature reduces the viscosity of the resin

making it easier to fill the mold. Increasing the pressure or injection rate increases the shear rate, which decreases the viscosity making it easier to fill the mold. Therefore, given a resin, machine and mold, there are three variables that can be used to fill out the mold:

- Injection/packing pressures
- Injection rate
- Melt temperature

The curves in Figure 37 indicate the relative temperature-pressure relationships for PE resins of varying melt indices. The higher the MFR or MI, the lower the injection pressure and/or the temperature required to fill a mold. Assuming the same mold filling characteristics (fill speed and fill time), cycle time and injection temperature, a high flow resin:

1. Will allow pressures to be reduced about 25% when the resin MI or MFR is doubled.
2. Will allow a decrease in melt temperature of about 70°F (40°C) when the resin MI or MFR is doubled.

The effect of a higher flow PE resin on temperature and pressure can be seen in Figure 38. Note that as the MI or MFR of the resin increases the possible reduction in temperature and/or pressure will become less.

However, the switch to a polyolefin with higher flow characteristics usually results in a loss of other properties such as resistance to stress cracking and impact strength, especially at lower temperatures.

Injection and/or packing pressures are typically the first settings adjusted by the processor because they have a quick response on mold fill. Increasing the pressures will help fill out the mold correcting for short shots and reducing or eliminating surface defects such as sink marks and ripples near the

gate. The downside of increasing pressure is the chance of trapping air in the cavity resulting in burn marks or of increasing the flash on the parting line due to the mold opening. Increasing injection pressures also pack resin more tightly into the cavity, which may reduce shrinkage, increase the gate temperature(s), and increase molded-in stress. The reduced shrinkage can lead to a part sticking in the cavity and also post-mold dimensional differences.

Increasing the injection rate(s) reduces the viscosity of the resin, which may reduce the amount of molded-in stress in the part. In addition, an increased injection rate may also yield a more uniform part temperature (due to faster introduction of material into the mold) which can reduce differential shrinkage (i.e. warpage) due to temperature variation. Increasing the injection rate(s) without a decrease in injection pressure can lead to flashing of the part. Changing the injection rate(s) also has a fast effect on part quality although it may take time to fine-tune the rate(s) for optimum quality. Excessive molded-in stress can lead to an increase in warpage and a decrease in impact strength and environmental stress cracking resistance.

Sometimes the injection rate and injection pressure are not independent variables; i.e. the machine is set with a maximum pressure and runs on injection rate settings which are set on screw position. This setup will allow the machine to vary the injection pressure based on the pressure needed to meet the rate set points. Conversely, the injection pressure can be specified based on screw position and the rate is allowed to vary. Some processors are now utilizing pressure sensors within the cavity to control the operation of the machine via cavity pressure. This is a new approach that is

Figure 37. Temperature-pressure relationships for polyethylene resins of several melt indices

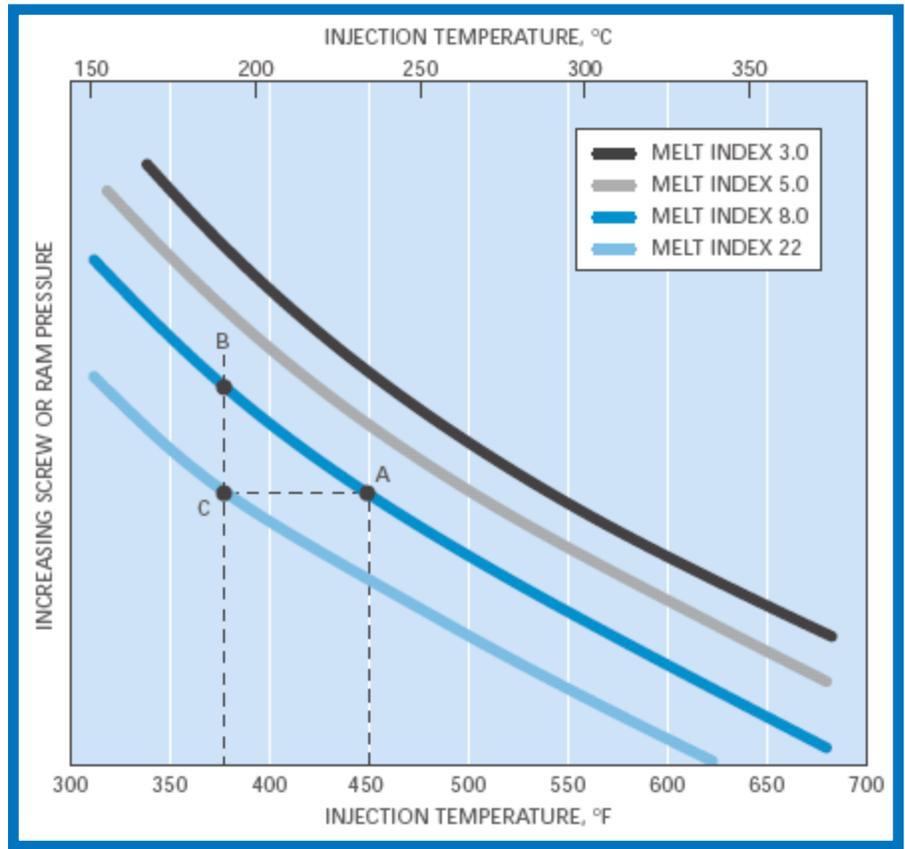
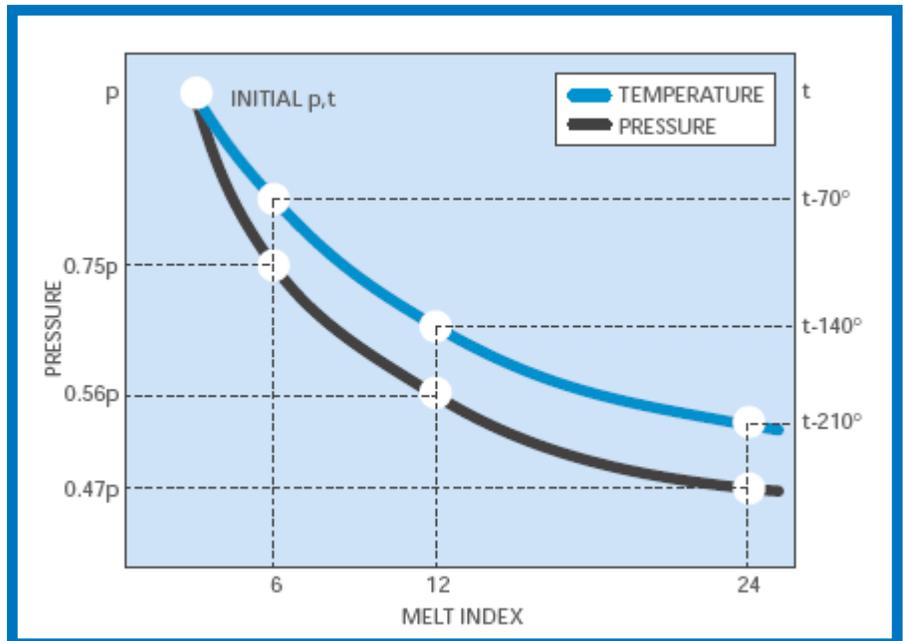


Figure 38. Effect of Melt index of polyethylene resin on injection temperature



gaining acceptance for molding parts with critical tolerances. It is also applicable to molds (such as syringes) where core shift is of concern.

The final way to control the viscosity of the resin is to adjust the melt temperature. An increase in temperature will decrease viscosity. Changing temperature settings yields a slower response than pressure or injection rate. High resin temperature can lead to degradation and require longer cooling time while low temperatures can lead to shot inconsistency, higher injection pressures, and excessive wear/damage to the screw and barrel.

When setting an injection rate or injection and packing pressure profile, the aim of the processor should be to provide a smooth delivery of material into the mold. A momentary slowing of the screw due to either the transfer from one step to another or too large of a step can result in a hesitation of the plastic flow front. Hesitation of the melt front can cause surface defects such as flow lines or tiger stripes, which may lead to poor weld and/or knitline strength. Therefore it is necessary to reduce the rates or pressures in a consistent manner to prevent flashing of the tool, potential core shift(s), and bottoming out of the screw.

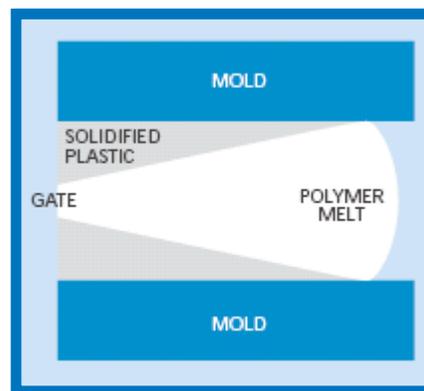
In general, >95% of the part weight(s) should be delivered during the injection step. The final part weight is achieved via the packing and holding step. Packing pressures are typically about half the level of the injection pressure and serve to achieve final part weight(s) and also to allow time for the gates to freeze off before plastication can begin for the next shot.

During the injection and packing steps, coolant (typically a mixture of ethylene glycol and deionized water if $\leq 45^{\circ}\text{F}$) is circulating

through the mold. The coolant takes heat out of the mold and therefore out of the part via conduction. Optimum cooling is achieved when the water is in turbulent flow. In general, an increase in coolant flow rate will remove more heat than a decrease in coolant temperature.

As resin flows into the mold, the material in contact with the mold surface solidifies very quickly forming a skin layer and an inner flow 'channel' through which material continues to flow (Figure 39). As more heat is taken from the plastic by the mold, the flow channel is reduced to the point that no more material will flow. The optimum process parameters should be chosen to allow complete mold fill and pack before the flow channel solidifies completely. Keeping the melt channel open allows for better packing of the extremities of the part. There are also differences in part temperature depending on proximity to the gate.

Figure 39.



Uniform cooling to the mold occurs when the coolant makes one pass through the mold (no looping or connecting of flow channels) and there is only one temperature controller. During even cooling, the gate area is always the hottest area of the part because throughout the injection and packing steps, molten material continues to flow through the gate.

The extremities of the part tend to have the lowest temperature since the polymer melt has transferred heat as it has flowed through the cavity.

Differences in part temperature can lead to differential shrinkage and therefore warpage. There are two ways to minimize temperature differentials either through differential cooling or an increase in injection rate.

Differential cooling involves directing coolant towards the gate area, which has the highest part temperature and away from the extremities where the part temperature is lowest. The typical method is to reduce the coolant flow to the cooling channels nearest the extremities and open up the valves to the channels nearest the gate. Directing the coolant flow from the hotter portions of the tool to the extremities may also be effective in some cases.

As covered previously, increasing the injection rate will shorten the injection times allowing for a more uniform part temperature.

Because the cooling step is usually the longest time period in the injection molding cycle, a cold mold temperature is generally recommended. Because of the wide range of mold and part designs, it is very difficult to specify mold temperatures. A typical range of mold surface temperatures for polyolefins is 70-125°F (20-50°C), which requires coolant temperatures of 32-50°F (0-10°C). Materials with lower melting temperatures, such as EVAs, will be at the lower end of the range while the higher melting materials, such as HDPE and PP will be at the higher end. Cold molds, however, tend to give a less glossy surface finish and will restrict the flow of resin within the mold. A cold

mold can also lead to a higher amount of molded-in stress within the part. Warmer mold temperatures will increase the gloss level and may also improve resin flow by constricting the melt channel less, at the expense of increased cycle times.

The injection molder and the mold design typically fix the amount of time needed for the mold to open, eject the part and then close again. Mold open time can be reduced by using only enough daylight to allow the part(s) to fall freely, reducing the amount of time required for air assists and also the number of times the ejector pins activate.

Shrinkage

Amorphous resins such as ABS, Polycarbonate, and Polystyrene have much lower shrinkage values than the polyolefins. The higher shrinkage of polyolefins is due to the fact that, in their molten state, they take up more volume than in the solid state because polyolefin resins are semi-crystalline. When the resin solidifies, the chains in the crystalline regions pack tightly together resulting in a reduction in volume. In general, the polyolefins can be ranked for shrinkage:

HDPE ≥ LLDPE ≥ LDPE ≥ PP

Once a resin has been selected, shrinkage can be controlled, to some extent, through mold design and processing conditions (Table 5). Studies on a test mold in which the thickness and gate area of a flat plaque can be varied, indicate the following:

- Shrinkage is reduced as part thickness decreases. The response to a thickness change is more pronounced with HDPE than PP.
- Shrinkage is reduced when the gate area is reduced.

Since the degree of shrinkage is partly a result of cooling, it can be reduced by molding at lower injection temperatures and running a colder mold. Packing the part more will also minimize shrinkage. This is done either by molding at moderate temperatures and high pressures or by molding at fairly high temperatures and moderate pressures. However, excessive temperature or pressure can result in flash.

Another means of reducing shrinkage is the use of higher pressure and longer packing time. This allows additional resin to flow into the mold as the material in the mold cools and shrinks, packing out the mold as much as possible but may also increase cycle time and higher molded-in stress.

Longer cooling time in the mold before ejection is especially useful whenever an inside dimension is critical. As the molded article cools and contracts around the core, the core will maintain the critical inside dimension of the part. Generous draft, or tapering, will allow easier part ejection. A longer cooling time will mean an increase in cycle time, therefore many molders will increase the mold cooling to reduce shrinkage.

Shrinkage is a time-dependent function. In general, a polyolefin part has achieved about 90% of its total shrinkage after 48 hours. Shrinkage can continue for several more days if the parts are packed hot and/or are stored in a warm warehouse. Parts that have shrunk after packaging typically exhibit 'nesting' problems if the parts are stacked inside each other.

Warpage

Warpage results from non-uniform shrinkage of the molded part caused by non-uniform cooling. When a part warps after being ejected, it is assuming its 'natural' shape by relieving the stresses forced upon it while being cooled in the mold. The problem, often a difficult one to solve, is to minimize the 'locked-in' stresses, which the part might later 'remember' and relieve during cooling to room temperature. In cases where parts are fixtured after ejection, subsequent exposure to higher temperatures may cause relaxation and warpage. Part designs incorporating significant differences in cross-sectional thickness are more prone to warpage than those with a more uniform thickness, due to higher residual temperatures in the thicker sections.

In addition to non-uniform cooling, locked-in stresses are generated in the mold by such operating conditions as excessive molding pressures, slow fill times, low backpressure, or too low a melt temperature.

There is no single, clear-cut remedy for warpage. Adjusting mold conditions, redesigning the part or the mold, switching to a material with a narrower MWD, or a combination of these may reduce the internal stresses. Generally, the least warpage occurs when the melt temperature is set at the maximum, the mold temperature is high, injection pressure is a minimum and the injection time is short (Table 5).

Molding at high temperatures allows the stresses induced during injection to be reduced before the part sets. Running a warm mold also allows the stresses to relax before the melt sets. Differential cooling between the mold halves is

Table 5. Some ways to reduce shrinkage and warpage in polyolefin injection moldings

	SHRINKAGE	WARPAGE
Molding conditions	Reduce cylinder temperature and lower mold temperature	Use high cylinder temperature and high mold temperature
	Lower temperatures near mold gating and sprue and entrance to molded part	Use low temperatures near mold gating and sprue and entrance to molded part
	Use minimum cooling at mold extremities	Use minimum cooling at mold extremities
	Moderate cylinder temperature and use very high injection pressure	Lower injection pressure
	Use fairly high cylinder temperature and moderate injection pressure	Use high cylinder temperature
	Use high injection pressure and extended injection time	Shorten injection time
	Use longer dwell time	Use longer dwell time
Mold design	Use proper location of sprue and gating	Use proper location of sprue and gating
Resin properties	Use lower density resin	Lower density resin of little importance
	Use higher MI or MFR resin	Use higher MI or MFR resin

often required to produce warp-free parts especially those having large, flat sections.

Injection and packing pressures should be adequate to permit easy fill but should not be set excessively high in order to allow some of the molded-in stresses to relax before the part sets.

Increasing the injection rate will decrease the injection time, which will allow the mold to fill faster before the extremities can cool too much. This gives the entire molding a chance to cool at a more uniform rate, which reduces the warpage.

Some of these remedies, such as high melt temperature or low injection pressure can increase the cycle time. Switching to a higher MFR/MI resin can offset the increase in time. Higher flow resins

will allow lower injection pressure, which can shorten the molding cycle. In addition, higher flow resins typically exhibit less “elastic memory” which can also reduce warpage. Lower density resins (for PE) are only slightly less susceptible to warpage than higher density resins.

Differences between flow and transflow shrinkage can result in warpage. HDPE is known to have a large difference between these two while PP is more balanced between the flow and transflow shrinkage.

Because both shrinkage and warpage are strongly influenced by the mold cooling patterns and part geometry (uniformity of thicknesses and flow patterns), it is very important that these be considered in the early stages of part and mold design.

Color dispersion and air entrapment

An effective means of improving dispersion and preventing air bubbles from getting into the mold with the melt is to use a breaker plate at the end of the barrel between the screw tip and the nozzle (Figure 40). Backpressure on the melt, in most cases, squeezes all the air out between the melting pellets and produces bubble-free moldings. The breaker plate may be ¼ inch (6.5 mm) thick and must be large enough to fit into the opening of the nozzle. The plate is drilled with 20- to-40, small diameter (1/32 inch or 0.8 mm) holes. Another option is to increase the back pressure on the screw making sure that it is not set too high so that the screw cannot recover in time for the next cycle. Increasing the backpressure and/or adding a breaker plate can also

aid in color dispersion and melt temperature homogeneity. When running at high processing temperatures, backpressure should be kept to a minimum to reduce degradation of the resin.

Part ejection and mold release

Mold release is affected by a number of factors. Some polyolefin molding resins exhibit better mold release than others do. It has been found that these resins have accompanying disadvantages, such as less gloss. Resins that develop a grainy or frosty surface release better than smooth, high-gloss moldings, such as those made from high MI grades or clarified polypropylene. However, even polyolefins of the same MI may vary in their mold-release properties. Resins sometimes are compounded with a mold-release additive.

Changing the mold design or one or more molding conditions without affecting the end properties of the molding usually can alleviate mold release problems. Mold release may be difficult if the mold is packed too tightly in an effort to reduce shrinkage. Frequently, a molded article sticks to the mold if the cycle time is too long and the molding shrinks to a core. In such cases, shortening the cooling time may improve mold release. On the other hand, the same problem can occur if the cycle is too short to allow the molding to shrink away from the cavity walls. In such cases, length-ening the cycle time may improve mold release. Draw stoning of the surfaces in the direction of mold opening may also alleviate this problem.

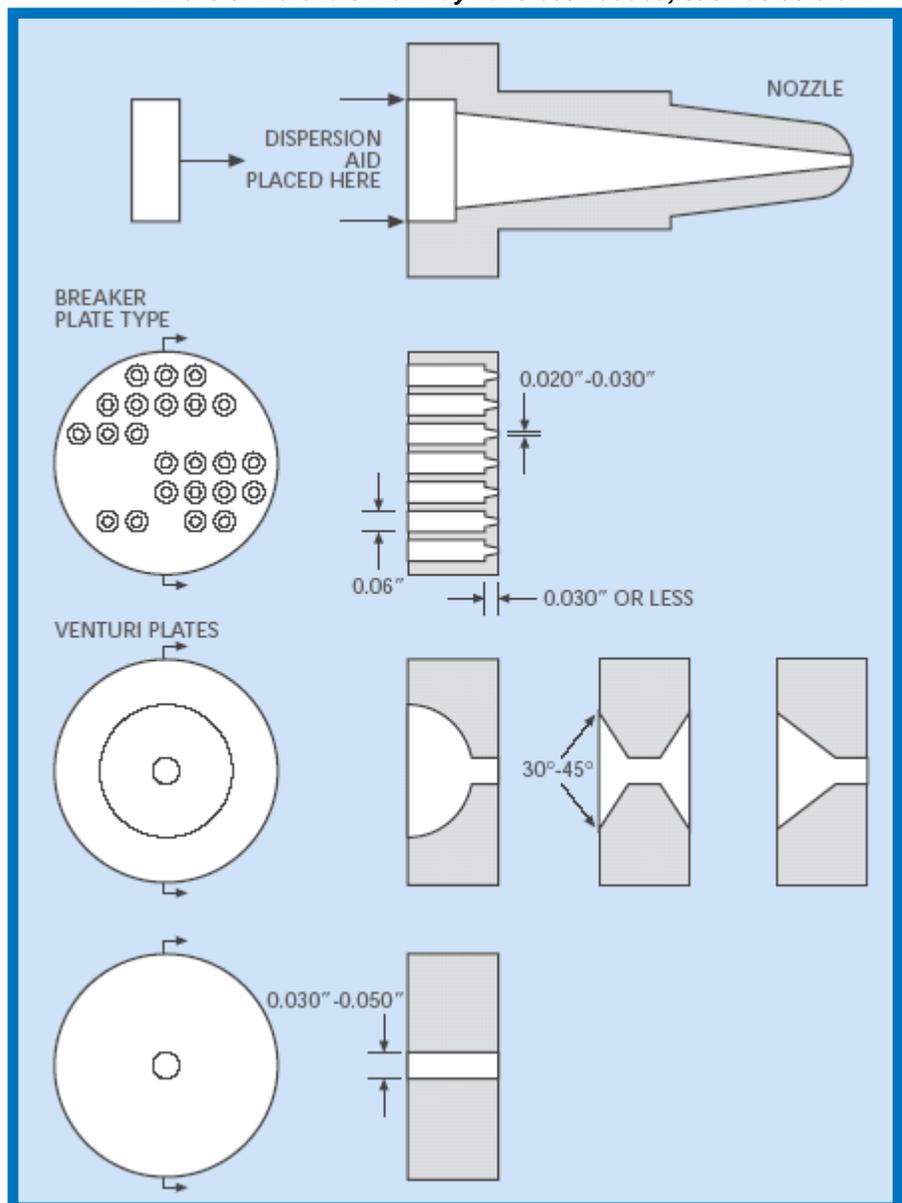
Mold release greatly depends on the degree of polish on the inside of the mold. Proper surface finishes inside a mold for a deep-draw item determines whether the part can be ejected easily or will

stick to the cavity or the mold core. Ejection pins may be used to move the molding from either the cavity or the core first.

Enough draft must always be provided, especially in deep-draw moldings, such as long containers. Reverse draft or even parallel sides should be avoided wherever possible. The draft allowance for

EVA moldings should be generous, as they tend to be stickier at part ejection temperatures. Normally, a 3- to 5-degree draft significantly assists in part ejection. On shallow draft molds, the use of stripper rings and/or air ejection may be necessary. Draft angles of less than one degree are not recommended and should be avoided unless dictated by part requirements.

Figure 40. Dispersion aids are inserts that may be placed in the nozzle of the injection molding machine to restrict the melt flow and improve mixing of the resin for a more homogenous melt temperature or mixture of materials that may have been added, such as colorant



Clarity

High melt temperature and low pressure are necessary to eliminate flow marks in molded parts. Clarity of molded parts also can be improved by lowering the mold temperature, especially in thin-walled sections when higher MI resins are used. This reduces the size of the crystals formed which, in turn, reduces the light diffraction.

Polypropylene random copolymers have greater clarity than PP homopolymers. The clarity of random copolymer resins can be further enhanced by the addition of clarifying agents. Optimum clarity for PP articles in the nominal 0.050 in. thickness range is obtained at melt temperatures of about 430°F (220°C) and mold temperatures about 50 to 80°F (10 to 25°C). Generally, high injection rates also enhance clarity. Highly polished tools are necessary for highest clarity.

Gloss

Surface gloss of the molding is affected by resin properties, the condition of the mold and molding conditions. The higher the MI or MFR of the polyolefin resin the greater the gloss of the molding. Further, higher density polyethylenes give higher gloss than lower density resins.

Highly polished molds are one of the most important factors for obtaining high-gloss parts. For polyethylenes, a warm mold gives better gloss than a cold mold. Gating also may contribute to obtaining a high-gloss part. Restrictive gating produces higher gloss because it keeps the temperature high as the melt is injected into the mold cavity. Highest gloss for polypropylene resins is obtained with a cold mold and a fast injection rate.

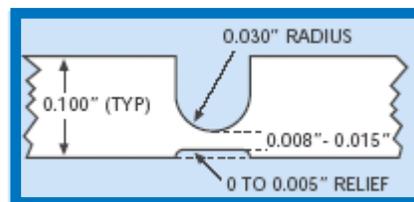
Polypropylene integral hinges

Polypropylene can be molded into a hinged part that can be flexed many cycles before failure. Lifetimes in excess of a million cycles have been measured for properly designed and manufactured hinged parts.

The hinged section must be thin enough to flex properly but thick enough to prevent tearing. Normal hinge thickness is 0.008-0.015 in. (0.2-0.38 mm). The greater thickness is necessary if the hinge requires strength or load bearing properties.

A typical cross-section of an integral hinge is shown in Figure 41. A shallow relief or clearance must be provided to prevent gathering and excessive stress when the hinge is in the closed position. Variations may be made on this design to achieve specific results.

Figure 41. Polypropylene integral hinge specifications



The gating into the part must be designed to allow the polymer to flow through the entire hinged section in a uniform and continuous fashion with the flow front perpendicular to the hinge. This arrangement ensures optimum hinge strength without delamination. It is preferable that all gates be on one side of the hinge to eliminate the possibility of weld lines; however, some designs are so complicated that this arrangement is not possible and

gates must be placed so a weld line does not occur in the hinge. If multiple gates are used, it is recommended that a section, slightly thicker than the wall thickness, be placed parallel to the hinge. This 'flow collector' will promote a uniform flow pattern across the hinge.

Molding conditions that lead to optimum hinge properties are a high melt temperature (typically 500°F to 525°F/260°C to 275°C), fast injection speed and a warm mold (120°F to 150°F/50°C to 65°C). In order to develop optimum properties, the hinge should be flexed several times immediately after removal from the mold. In some applications such as multicavity, hinged closures, it is not possible to perform this flexing. However, if the hinge is properly designed, it will still perform adequately for the requirements of the application. All polypropylenes can be used for living hinges but the most optimum hinges are achieved with homopolymer PP followed by random copolymer and impact PP. Acceptable hinges may be formed from PP impact copolymers but there is some potential for delamination in the hinge area.

It is also possible to produce hinges utilizing secondary operations. One post-forming procedure involves a heated steel die at about 425°F (220°C) which is forced into the molded part at 50-100 psi (345-690 kPa) of pressure. A rolling, heated die can also be used; this process is often referred to as 'coining' a hinge. If the polymer thickness is reduced to 0.005 to 0.015 in. (0.13-0.38 mm) a satisfactory hinge results. This technique may be used to put hinges in very large or complex parts.

Appendix 1

Some terms pertaining to injection molding

Antioxidant: Additive used to help protect plastics from degradation through sources such as heat, age, chemicals, stress, etc.

Antistatic Agent: Additive used to help eliminate or lessen static electricity from the surface of the plastic part.

Aspect Ratio: Ratio of total flow length to average wall thickness.

Back Pressure: The pressure applied to the plastic during screw recovery. By increasing back pressure, mixing and plasticating are improved; however, screw recovery rates are reduced.

Backing Plate: A plate used as a support for the mold cavity block, guide pins, bushings, etc.

Boss: Protuberance on a plastic part designed to add strength, facilitate alignment, provide fastening, etc.

Breaker Plate: See Figure 40.

Cavity: The space inside a mold into which material is injected. (Figure 42).

Charge: The measurement or weight of material necessary to fill a mold during one cycle.

Clamping Plate: A plate fitted to a mold and used to fasten the mold to a platen (Figure 42).

Clamping Pressure: The pressure applied to the mold to keep it closed during a cycle, usually expressed in tons.

Closed-loop Control: System for monitoring complete, injection-molding-process conditions of temperature, pressure and time, and automatically making any changes required to keep part production within preset tolerances.

Co-injection Molding: A special multimaterial injection process in which a mold cavity is first partially filled with one plastic and then a second shot is injected to encapsulate the first shot.

Cooling Channels: Channels located within the body of a mold through which a cooling medium is circulated to control the mold surface temperature.

Clarifiers: Additive used in polypropylene random copolymers to improve clarity.

Cushion: Extra material left in barrel during cycle to try and ensure that the part is packed out during the hold time.

Cycle: The complete sequence of operations in a process to complete one set of moldings. The cycle is taken at a point in the operation and ends when this point is again reached.

Daylight Opening: The maximum distance between the stationary and moving platens of the clamp unit in the fully open position.

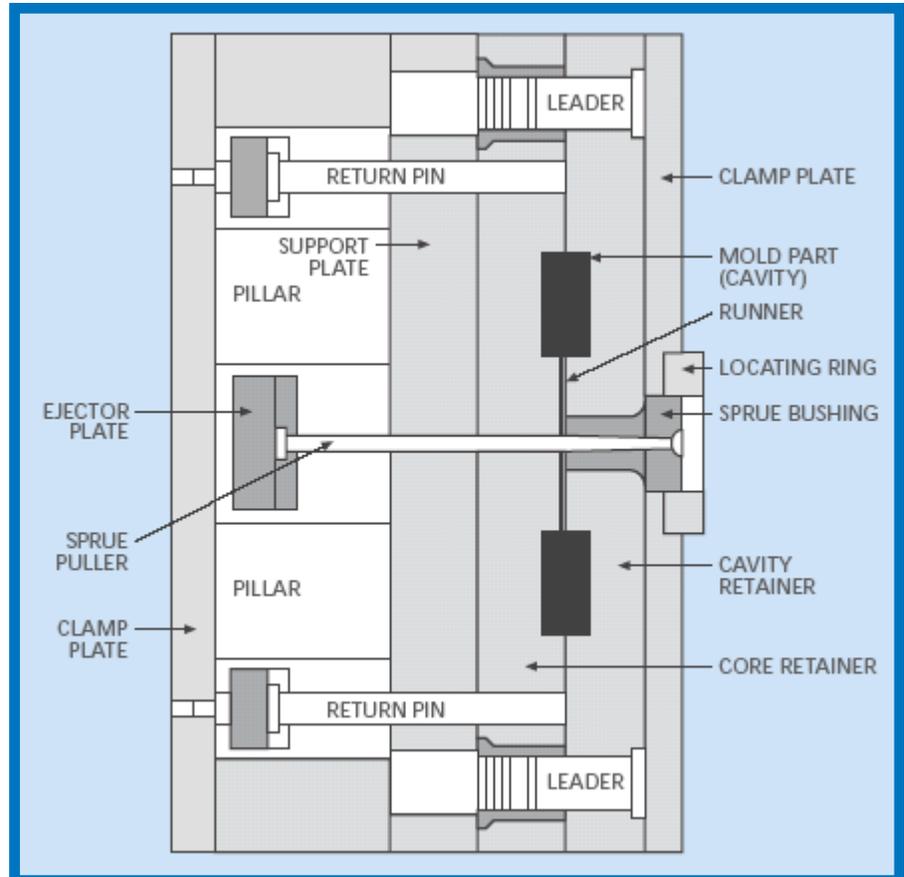
Delamination: When the surface of a finished part separates or appears to be composed of layers. Strata or fish-scale-type appearance where the layers may be separated.

Diaphragm Gate: Used in symmetrical cavity filling to reduce weld-line formations and improve filling rates.

Direct Gate: The sprue that feeds directly into the mold cavity.

Dispersion Aids: Perforated plates placed in the plasticator nozzle to aid in mixing or dispersing colorant as it flows through the perforations (Figure 40).

Figure 42. Schematic showing a typical injection mold with some of the points identified



Draft: The degree of taper of a mold-cavity sidewall or the angle of clearance designed to facilitate removal of parts from a mold.

Drooling: The extrusion or leakage of molten resin from a plasticator nozzle or nozzle sprue-bushing area while filling or shooting the mold.

Dwell: A pause in the applied pressure to a mold during the injection cycle just before the mold is completely closed. This dwell allows any gases formed or present to escape from the molding material.

Ejector Pins: Pins that are pushed into a mold cavity from the rear as the mold opens to force the finished part out of the mold. Also called knockout pins.

Ejector Return Pins: Projections that push the ejector assembly back as the mold closes. Also called surface pins or return pins.

Ejector Rod: A bar that actuates the ejector assembly when the mold opens.

Family Mold: A multi-cavity mold where each of the cavities forms one of the component parts of an assembled finished part.

Fan Gate: A gate used to help reduce stress concentrations in the gate area by spreading the opening over a wider area. Less warping of parts can usually be expected by the use of this type of gate.

Fill: The packing of the cavity or cavities of the mold as required to give a complete part or parts that are free of flash.

Fin: The web of material remaining in holes or openings in a molded part which must be removed for final assembly.

Flash: Extra plastic attached to a molding usually along the mold parting line.

Flow: A qualitative description of the fluidity of a plastic material during the process of molding. A measure of its moldability generally expressed as melt flow rate or melt index.

Flow Line: Marks visible on the finished items that indicate the direction of the flow of the melt into the mold.

Flow Marks: Wavy surface appearances on a molded part caused by improper flow of the melt into the mold.

Gate: An orifice through which the melt enters the mold cavity.

Hob: A master model in hardened steel. The hob is used to sink the shape of a mold into a soft metal block.

Homopolymer: Plastic that results by the polymerization of a single monomer.

Hopper Dryers: Auxiliary equipment that removes moisture from resin pellets.

Hopper Loader: Auxiliary equipment for automatically loading resin pellets into machine hopper.

Hot-runner Mold: A mold in which the runners are insulated from the chilled cavities and are kept hot. Hot-runner molds make parts that have no scrap.

Injection Pressure: The pressure on the face of the injection screw or ram when injecting material into the mold, usually expressed in psi.

Insulated Runner: See hot-runner mold.

Izod Impact Test: Test to determine impact strength of a sample by holding a sample bar at one end and broken by striking. Sample specimen can be either notched or unnotched.

Jetting: A turbulent flow in the melt caused by an undersized gate or where a thin section rapidly becomes thicker.

Jig: A tool for holding parts of an assembly during the manufacturing process.

Knit Lines: See weld lines.

Knockout Pins: A rod or device for knocking a finished part out of a mold.

L/D Ratio: A term used to help define an injection screw. This is the screw length-to-diameter ratio.

Melt Flow Rate: A measure of the molten viscosity of a polymer determined by the weight of polymer extruded through an orifice under specified conditions of pressure and temperature. Particular conditions are dependent upon the type of polymer being tested. MFR usually is reported in grams per 10 minutes. Melt flow rate defines the flow of a polypropylene resin. An extrusion weight of 2160 grams at 446°F (230°C) is used.

Melt Index: Term that defines the melt flow rate of a polyethylene resin. An extrusion weight of 2160 grams at 310°F (190°C) is used.

Mold Changer: An automated device for removing one mold from a machine and replacing it with another mold.

Mold Frame: A series of steel plates which contain mold components, including cavities, cores, runner system, cooling system, ejection system, etc.

Mold-temperature-control Unit: Auxiliary equipment used to control mold temperature. Some units can both heat and cool the mold. Others, called chillers, only cool the mold.

Moving Platen: The platen of an injection molding machine that is moved by a hydraulic ram or mechanical toggle. (Figure 20-21)

Multi-cavity Mold: A mold having two or more impressions for forming finished items in one machine cycle.

Multi-material Molding: The injection of two-or-three materials, in sequence, into a single mold during a single molding cycle. The injection molding machine is equipped with two-or-three plasticators. (See also co-injection)

Nest plate: A retainer plate in the mold with a depressed area for cavity blocks.

Non-return Valve: Screw tip that allows for material to flow in one direction and closes to prevent back flow and inject material into the mold.

Nozzle: The hollow-cored, metal nose screwed into the injection end of a plasticator. The nozzle matches the depression in the mold. This nozzle allows transfer of the melt from the plasticator to the runner system and cavities.

Nucleating Agent: Additive used with polypropylene to increase crystallization rate by providing additional sites for crystal growth.

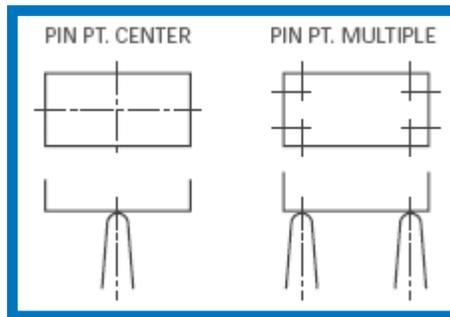
Orange Peel: A surface finish on a molded part that is rough and blotchy. Usually caused by moisture in the mold cavity.

Packing: The filling of the mold cavity or cavities as full as possible without causing undue stress on the molds or causing flash to appear on the finished parts.

Part Picker: An auxiliary unit usually mounted on fixed platen, which reaches into the open mold to grab parts and remove them prior to next molding cycle. Also called a robot, the device is used when you do not want to drop parts from mold upon ejection.

Parting Line: On a finished part, this line shows where the two mold halves met when they were closed.

Pinpoint Gate: A restricted gate of 0.030 in or less in diameter, this gate is common on hot-runner molds.



Piston: See ram.

Plasticate: To soften by heating and mixing.

Plasticator: The complete melting and injection unit on an injection molding machine.

Platens: The mounting plates of a press on which the mold halves are attached.

Plate-out: The blooming of additives onto machinery during processing of plastics.

Plunger: See ram.

Pressure Pads: Reinforcements of hardened steel distributed around the dead areas in the faces of a mold to help the land absorb the final pressure of closing without collapsing.

Purging: The forcing one molding material out of the plasticator with another material prior to molding a new material. Special purging compounds are used.

Ram: The forward motion of the screw in the plasticator barrel that forces the melt into the mold cavity.

Recovery Time: The length of time for the screw to rotate and create a shot.

Restricted Gate: A very small orifice between runner and cavity in an injection mold. When the part is ejected, this gate readily breaks free of the runner system.

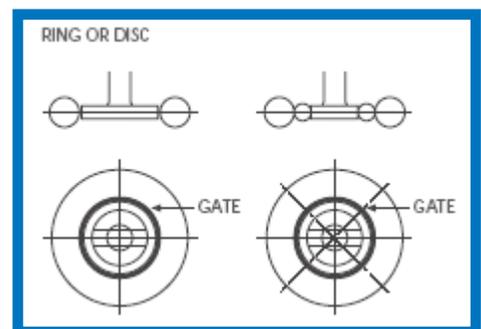
Generally, the part drops through one chute and the runner system through another leading to a granulator and scrap-reclaim system.

Retainer Plate: The plate on which demountable pieces, such as mold cavities, ejector pins, guide pins and bushings are mounted during molding.

Retractable Cores: Used when molding parts in cavities not perpendicular to the direction in which the part is ejected from the mold. The cores are automatically pulled from the mold prior to the mold opening and reinserted when the mold closes again and prior to injection.

Rib: A reinforcing member of a molded part.

Ring Gate: Used on some cylindrical shapes. This gate encircles the core to permit the melt to first move around the core before filling the cavity.



Robot: Automated devices for removing parts upon ejection from an open mold rather than letting the parts drop. Also see parts picker. Robots also can perform secondary functions, such as inspection, degating, precise placement of parts on a conveyor, etc.

RMS Roughness: A measure of the surface roughness/smoothness of a material. The root mean square (RMS) average of the "peaks and valleys" of a surface is determined using a Profilometer. The lower the number, the smoother the surface: a reading of one or two would be a very polished and smooth surface.

Rockwell Hardness: A measure of the surface hardness of a material. A value derived from the increase in depth of an impression as the load of a steel indenter is increased from a fixed minimum value to a higher value and then returned to the minimum value. The values are quoted with a letter prefix corresponding to a scale relating to a given combination of load and indenter.

Runner: The channel that connects the sprue with the gate for transferring the melt to the cavities.

Runnerless molding: See hot-runner mold.

Screw Travel: The distance the screw travels forward when filling the mold cavity.

Short Shot: Failure to completely fill the mold or cavities of the mold.

Shot: The complete amount of melt injected during a molding cycle, including that which fills the runner system.

Shot Capacity: Generally based on polystyrene, this is the maximum weight of plastic that can be displaced or injected by a single injection stroke. Generally expressed as ounces of polystyrene.

Shrinkage: The dimensional differences between a molded part and the actual mold dimensions.

Side Bars: Loose pieces used to carry one or more molding pins and operated from outside the mold.

Side-draw Pins: Projections used to core a hole in a direction other than the line of closing of a mold and which must be withdrawn before the part is ejected from the mold. See also Retractable Cores.

Silver Streaks: See splay marks.

Single-cavity Mold: A mold having only one cavity and producing only one finished part per cycle.

Sink Mark: A shallow depression or dimple on the surface of a finished part created by shrinkage or low fill of the cavity.

Slip Agent: Additive used to provide lubrication during and immediately following processing of plastics.

Slip plane: Marks evident in or on finished parts due to poor welding or shrinking upon cooling.

Spiral Flow: Test performed by injection molding a sample into a spiral mold and used to compare the processability of different resins.

Splash Marks: See splay marks.

Splay Marks: Marks or droplet-type imperfections on the surface of the finished parts that may be caused by the splaying of the melt through the gates and into the cool cavity where they set up.

Split-ring Mold: A mold in which a split cavity block is assembled in a channel to permit the forming of undercuts in a molded piece. These parts are ejected from the mold and then separated from the piece.

Sprue Bushing: A hardened-steel insert in the mold that accepts the plasticator nozzle and provides an opening for transferring the melt.

Sprue Gate: A passageway through which melt flows from the nozzle to the mold cavity.

Sprue Lock: The portion of resin retained in the cold-slug well by an undercut. This lock is used to pull the sprue out of the bushing as the mold opens. The sprue lock itself is pushed out of the mold by an ejector pin.

Sprue: The feed opening provided in injection molding between the nozzle and cavity or runner system.

Stack Molds: Two or more molds of a similar type that are positioned one behind the other to allow for additional parts to be manufactured during a cycle.

Stationary Platen: The large front plate of an injection molding press to which the front plate of the mold is secured. This platen does not move during normal operation.

Stress Cracking: There are three types of stress cracking:

1. Thermal stress cracking is caused by prolonged exposure of the part to elevated temperatures or sunlight.
2. Physical stress cracking occurs between crystalline and amorphous portions of the part when the part is under an internally or externally induced strain.
3. Chemical stress cracking occurs when a liquid or gas permeates the part's surface.

All of these types of stress cracking have the same end result: the splitting or fracturing of the molding.

Striations: Marks evident on the molded-part surfaces that indicate melt flow directions or impingement.

Stringing: Occurs between the finished part and the sprue when the mold opens and the melt in this area has not cooled sufficiently.

Stripper Plate: A plate that strips a molded piece from core pins or force plugs. The stripper plate is set into operation by the opening of the mold.

Structural Foam Molding:

Process for making parts that have solid outer skin and foamed core.

Submarine Gate: A gate where the opening from the runner into the mold cavity is located below the parting line. Also called a tunnel gate.

Suck-back: When the pressure on the sprue is not held long enough for the melt to cool before the screw returns. Some of the melt in the cavities or runner system may expand back into the nozzle and cause sink marks on the finished part.

Tab Gate: A small removable tab about the same thickness as the molded item, but usually perpendicular to the part for easy removal.

Tie-bar Spacing: The space between the horizontal tie-bars on an injection molding machine. Basically, this measurement limits the size of molds that can be placed between the tie-bars and into the molding machine.

Toggle: A type of clamping mechanism that exerts pressure by applying force on a knee joint. A toggle is used to close and exert pressure on a mold in a press.

Tunnel Gate: See submarine gate.

Undercut: A protuberance or indentation that impedes withdrawal from a two-piece rigid mold.

Valve Gating: A type of gate where a pin is held in the gate or channel by spring tension. As the injection stroke moves forward, this gate compresses the plastic in the runner. When this pressure build-up is sufficient to overcome the spring tension, the pin is then pushed back (pulled) and the fast decompression of the melt fills the cavity at extremely high speed.

Vent: A shallow channel or opening cut in the cavity to allow air or gases to escape as the melt fills the cavity.

Vented Barrel: Special plasticator unit with a vent port over the compression section of the screw to permit escape of gases prior to injecting melt into mold. Often used when molding moisture-sensitive resins.

Vertical Flash Ring: The clearance between the force plug and the vertical wall of the cavity in a positive or semi-positive mold. Also the ring of excess melt which escapes from the cavity into this clearance space.

Warping: Dimensional distortion in a molded object.

Weld Line: Marks visible on a finished part made by the meeting of two melt-flow fronts during molding.

Wisps: Similar to stringing but smaller in size. These also may occur as slight flashing when the mold is overpacked or forced open slightly. Mold-parting-line wear or misalignment can also cause wisps.

Appendix 2 Metric Conversion Guide

Appendix 3: Metric Conversion Guide

<u>To Convert From</u>	<u>To</u>	<u>Multiply By</u>
<u>Area</u>		
square inches	square meters	645.2
square millimeters	square inches	0.0016
square inches	square centimeters	6.452
square centimeters	square inches	0.155
square feet	square meters	0.0929
square meters	square feet	10.76
<u>Density</u>		
pounds/cubic inch	grams/cubic centimeter	27.68
grams/cubic centimeter	pounds/cubic inch	0.036
pounds/cubic foot	grams/cubic centimeter	0.016
grams/cubic centimeter	pounds/cubic foot	62.43
<u>Energy</u>		
foot-pounds	Joules	1.356
Joules	foot-pounds	0.7375
inch-pound	Joules	0.113
Joules	inch-pounds	8.85
foot-pounds/inch	Joules/meter	53.4
Joules/meter	foot-pounds/inch	0.0187
foot-pounds/inch	Joules/centimeter	0.534
Joules/centimeter	foot-pounds/inch	1.87
foot-pounds/square inch	kilo Joules/square meter	2.103
kilo Joules/square meter	foot-pounds/square inch	0.4755
<u>Length</u>		
Mil	millimeter	0.0254
millimeter	mil	39.4
inch	millimeter	25.4
millimeter	inch	0.0394
<u>Output</u>		
pounds/minute	grams/second	7.56
grams/second	pounds/minute	0.132
pounds/hour	kilograms/hour	0.454
kilograms/hour	pounds/hour	2.20
<u>Power</u>		
kilowatts	horsepower(metric)	1.34
horsepower (metric)	kilowatts	0.76

<u>To Convert From</u>	<u>To</u>	<u>Multiply By</u>
<u>Pressure</u>		
pounds/square inch (psi)	kilopascals (kPa)	6.895
kilopascals (kPa)	pounds/square inch (psi)	0.145
pounds/square inch (psi)	bar	0.0689
bar	pounds/square inch (psi)	14.5
<u>Temperature</u>		
°F	°C	(°F-32)/1.8)
°C	°F	1.8 x °C +32
<u>Thermal Conductivity</u>		
Btu-in/hr, sq. ft., °F	W/(m-°K)	0.1442
W / (m-°K)	Btu-in/hr,sq ft, °F	6.933
<u>Thermal Expansion</u>		
inches/inch, °F	meters/meter, °C	1 . 8
meters/meter, °C	inches/inch, °F	0.556
<u>Viscosity</u>		
poise	Pa-sec.	0.1
Pa-sec	poise	10
<u>Volume</u>		
cubic inch	cubic centimeter	16.39
cubic centimeter	cubic inch	0.061
cubic foot	cubic decimeter	23.32
cubic decimeter	cubic foot	0.0353
<u>Weight</u>		
ounce	gram	28.35
kilogram	ounce	0.0353
pound	kilogram	0.4536
kilogram	pound	2.205
ton (US)	ton (metric)	0.907
ton (metric)	ton (US)	1.102

Appendix 3:

Abbreviations

ASTM	American Society for Testing and Materials
Btu	British thermal unit
cpm	Cycles per minute
cps	Cylinder
deg	Cycles per second
deflec	Deflection
deg	Degree (angle)
DTUL	Deflection temperature under load
E	Modulus of elasticity
elong	Elongation
EMA	Ethylene-vinyl acetate copolymer
ESCR	Environmental stress cracking resistance
EVA	Ethylene vinyl acetate copolymer
flex	Flexural
FR	Flame retardant
FRP	Fiber reinforced plastic
g	Gram
HDPE	High density polyethylene
HDT	Heat deflection temperature
HMW	High molecular weight
IMM	Injection molding machine
imp	Impact
J	Joule
K	Kelvin
kpsi	1000 pounds per square inch
L/D	Length to diameter ratio of screw
lb_f	Pound-force
LDPE	Low density polyethylene
LLDPE	Linear low density polyethylene
MD	Machine direction
MDPE	Medium density polyethylene
MFR	Melt flow rate
MI	Melt index
mod	Modulus
mol%	Mole percent
MW	Molecular weight
N	Newton
PE	Polyethylene
PP	Polypropylene
pphr	Parts per hundred resin
ppm	Parts per million
psi	Pounds per square inch
RH	Relative humidity
rpm	Revolutions per minute
sp gr	Specific gravity
SPE	Society of Plastics Engineers
SPI	The Society of the Plastics Industry
ten	Tensile
T_g	Glass transition temperature (crystalline polymers)
T_m	Melt temperature (amorphous polymers)
UL	Underwriter's Laboratories
ult	Ultimate
UV	Ultraviolet
yield	Yield

Appendix 4

ASTM test methods applicable to polyethylene

PROPERTY	ASTM METHOD	ISO METHOD
Ash Content	D 5630	ISO 3451-1, 6427
Deflection Temperature	D 648	ISO 75
Density	D 1505	ISO 1183
Dielectric Constant	D 150 or D 1531	—
Dissipation Factor	D 150 or D 1531	—
Environmental Stress Crack Resistance	D 1693	ISO 4599
Flexural Modulus (secant or tangent)	D 790	ISO 178
Flexural Strength	D 790	ISO 178
Flow Rates using extrusion rheometry	D 1238	ISO 1133
Hardness, Rockwell	D 785	ISO 2039-2
Hardness, Shore	D 2240	—
Impact Strength	D 256 for notched (Izod)	ISO 180
	D 4812 for unnotched	ISO 180
Impact Strength, Gardner	D 5420	—
Impact Strength, falling Dart (Tup impact)	D 5628	—
Impact Strength (multi-axial instrumented)	D 3763	ISO 5503-2
Impact, Tensile	D 1822	ISO 8256
Low Temperature Brittleness	D 746	ISO 974
Melt Flow Rate	D 1238	ISO 1133
Melt Index	D 1238	ISO 1133
Rheological properties using capillary rheometer	D 3835	ISO 11443
Shrinkage, mold	D 955	ISO 294-4
Specific Gravity	D 792	ISO 1183
Surface Resistivity	D 257	IEC 93
Tensile Modulus	D 638	ISO 527-1, 2
Tensile Strength	D 638	ISO 527-1, 2
Thermal Conductivity	C 177	—
Thermal Expansion	D 696	—
Torsional Expansion	D 1043	ISO 458-1
Vicat Softening Point	D 1525	ISO 306
Volume Resistivity	D 257	IEC 93
Water Absorption	D 570	ISO 62

None: Caution should be exercised when comparing data obtained using ASTM and ISO test procedures. In many cases, the data are not equivalent due to differences in specimen preparation, geometry, and/or testing procedures.

Appendix 5

Injection Molding Problems, Causes and Solutions

PROBLEMS	POSSIBLE CAUSE	SUGGESTED SOLUTIONS
Black Specks	Degraded plastic flaking off barrel, check ring, or screw	Purge heating barrel Reduce screw recovery and /or dwell time Clean screw and barrel
Black Streaks	Resin contamination	Examine resin for contamination
	Rear cylinder temp. too low	Possible air entrapment, raise temp
	Resin dead spots in melt	Examine nozzle seat, plasticator cylinder walls and valves for dead spots, surface roughness Polish and streamline all areas with which melt comes into contact
	Degradation or resin	Lower melt temperatures Lower back pressure Change gate size
Brittleness	Mold temperature too high	Decrease mold temperature Check mold cooling channels for blockage
	Resin too cold	Increase barrel temperature Check thermocouples
	Improper part design	Increase radii at corners, notch or on thread
Bubbles	Degradation or resin	Decrease melt temperature Increase injection pressure Increase dwell time
	Air entrapment	Increase back pressure Increase feed hopper heat zone
Burning	Improper venting	Vent cavity at point of air weldline entrapment Increase vent size
Chatter	Poor melt flow	Increase stock temp., decrease injection pressure Increase mold temp., decrease gate size
	Low melt index	Try above methods. If not applicable, obtain higher melt index resin
Clear spots	Unplasticized pellets	Improve machine's plasticizing capacity by adding special nozzle Increase cylinder temp. Increase cycle time Use intensive mixer nozzle Increase backpressure Change RPM; use higher compression screw Increase feed zone temp.
Color dispersion	Poor homogenization	Increase back pressure Increase stock temp. Use intensive mixer in nozzle Use higher flow resins in concentrate

Appendix 5

Injection Molding Problems, Causes and Solutions

PROBLEMS	POSSIBLE CAUSE	SUGGESTED SOLUTIONS
Delamination (skinning)	Contamination	Examination resin for contamination Purge cylinder Check for soap or other additives in colorant Check resin used to make color concentrate
	Resin temp. too low	Increase stock temp.
	Non-uniformity of resin temp.	Use higher back pressure Change RPM Use higher compression screw
	Mold temperature too high	Decrease mold temp.
Flow lines	Poor homogenization	Increase stock temp. Increase injection pressure Increase nozzle temp.
	Mold temp. too low or unbalanced	Increase mold temp. Check mold for cold spots and balance
Gate brittleness	Mold temperature too high	Decrease mold temp
	Inadequate cooling in gate	Improve cooling in gate to prevent heat build-up
	Over packing	Reduce dwell time Reduce feed Use minimum injection time
	Gate section of part too thin	Increase gate thickness to about 20% greater than thickest section of part
	Resin too cold	Increase stock temp. Increase RPM Use higher compression screw
	Non-uniform resin temp.	Increase cycle time Use intensive mixer in nozzle Increase back pressure
Mold flashing	Resin too hot	Reduce stock temp. Reduce nozzle temp. Shorten cycle time Increase gate size Reduce back pressure Change RPM
	Mold temp too high	Lower mold temp
	Improper mold surface seal	Check mold surface Check closing unit linkage or hydraulic psi Check for hydraulic leakage
	Injection pressure too high	Decrease injection pressure, time, velocity
	Clamp pressure set too low or projected area too large for clamp pressure of unit	Adjust clamp pressure to maximum Use higher flow resin Restrict gates Use larger machine

Appendix 5

Injection Molding Problems, Causes and Solutions

PROBLEMS	POSSIBLE CAUSE	SUGGESTED SOLUTIONS
Over packing	Excessive resin flow	Decrease stock temp. Decrease injection pressure Decrease dwell time Decrease nozzle temp. Increase gate size
Poor surface	Poor mold surface	Prepare mold surface as needed to obtain desired finish
	Material or mold too cold	Increase stock temp. Increase cycle time Increase mold temp
	Air entrapment	Open vents Add vents as needed
	Melt flow too low	Use higher flow resin
	Mold design	Relocate gates to change flow pattern andn reduce turbulent flow in mold
	Injection pressure too high	Reduce injection pressuree
Poor weld lines	Material too cold	Increase stock temp. Increase back pressure Increase nozzle temp Change RPM
	Injection speed too low	Increase injection pressure Increase injection speed
	Mold design	Relocate gates to change flow pattern Increase thickness in weld line area Add rib Enlarge gate size
	Contamination or poorly dispersed colorant	Check system for contamination Use color concentrate Use intensive mixer or dispersion and higher flow resin in concentrate
	Color shifting (mold alignment)	Check for core and mold alignment and correct
Silver streaking	Melt flow too high	Decrease stock temp.
	Melt flow too slow	Increase injection pressure Increase dwell time Increase mold temp.
Sinks and voids	Not enough cushion	Increase feed (do not overpack) Increase dwell time Increase mold temp.
	Material temp. too high	Decrease stock temp.
	Mold design	Decrease nozzle temp. Enlarge runners and gates to reduce freeze-off Locate gates near thick sections
	Parts cool too rapidly	Heat mold up Decrease cycle
	Rib sections too wide	Decrease thickness to about 80% or less of adjoining wall sections
	Mold temp. unbalanced and too high	Balance mold cooling Lower mold temp.

Appendix 5

Injection Molding Problems, Causes and Solutions

PROBLEMS	POSSIBLE CAUSE	SUGGESTED SOLUTIONS
Short shots	Injection pressure too low	Increase injection pressure. Increase resin feed Increase dwell time Check for flash (damaged mold) Check non-return valve
	Stock or mold temp. too low	Prepare mold surface as needed to obtain desired finish
	Material or mold too cold	Increase stock temp. Increase nozzle temp. Increase cycle time Increase mold temp. Increase back pressure Increase RPM
	Insufficient cushion	Increase feed (do not overpack) Increase dwell time Increase mold temp.
Shrinkage	Ejected part too hot	Increase cycle time Lower stock temp Lower nozzle temp. Increase injection pressure Lower mold temp. Increase dwell time
Warpage	Mold design	Large flat areas Non-uniform wall thickness Inadequate ribbing Gates not properly located Cooling channels not properly located
	Ejected part too hot	Increase cycle time Reduce stock temp. Jig part Reduce mold temp. Reduce nozzle temp. Enlarge gate size Reduce back pressure
	Inadequate feed	Increase feed Increase dwell time, but don't overpack
	Improper cycle set up	Increase mold clamp time Increase injection speed
	Parts cool too rapidly	Heat mold up Decrease cycle

Appendix 6

Comparison between ASTM and ISO

Sample Preparation and Test Procedures

SAMPLE PREPARATION

For polyethylene, compression molding is the preferred sample preparation by ASTM.

For polypropylene, injection molding is the preferred sample preparation by both ASTM and ISO.

FLEXURAL MODULUS

For ASTM, the recommended test specimen is 127 mm by 12.7 mm by 3.2 mm.

For ISO, the recommended test specimen is 80 mm by 10 mm by 4.0 mm.

IZOD IMPACT

For ASTM, the recommended test specimen is 64 mm by 12.7 mm by 3.2 mm.

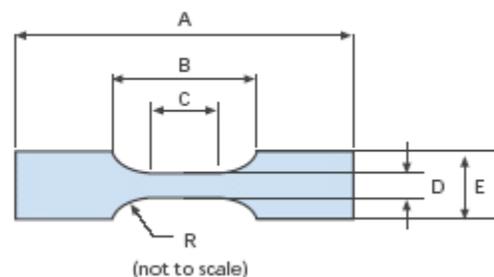
For ISO, the recommended test specimen is 80 mm by 10 mm by 4.0 mm.

For notched Izod Impact testing, both ASTM and ISO used a notch at an angle of 45° with a radius of curvature of 0.25 mm.

TENSILE IMPACT

The test specimen geometries, as shown in the figure, are as follows:

	ASTM	ISO
A, mm	63.5	60
B, mm	25.4	25
C, mm	9.5	10
D, mm	3.2	3
E, mm	12.7	10
R (Radius of Curvature), mm	12.7	10



TENSILE IMPACT

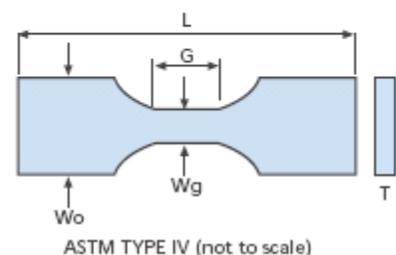
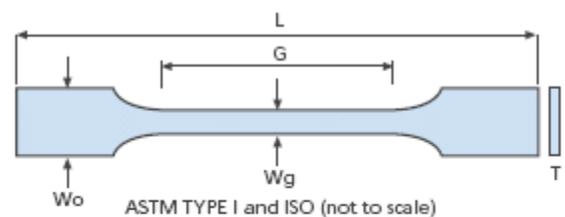
For ASTM, an end-gated, single-cavity injection mold with dimensions of 127 mm by 12.7 mm by 3.2 mm is recommended.

For ISO, a fan-gated, two-cavity injection mold with dimensions of 60 mm by 60 mm by 2 mm is recommended.

TENSILE STRENGTH AND ELONGATION

The test specimen geometries, as shown in the figure, are as follows:

	ASTM TYPE I	ISO	ASTM TYPE IV
L, overall length	165 mm	170 mm	115 mm
G, gage length	50	80	25
Wo, overall width	19	20	19
Wg, gage width	13	10	6
T, thickness	3.2	4.0	1.9



Appendix 7

Comparison between Compression Molded and Injection Molded Sample Preparation for HDPE*

	Compression Molded Specimen	Injection Molded Specimen
1% Secant Modulus, psi	163,000	133,000
Tensile Strength at Break, psi	4,130	3,700
Tensile Strength at Yield, psi	3,910	3,320
Elongation at Break, %	1,500	830
Elongation at Yield, %	7	9
Tensile Impact, ft.lbs./in.2	60	45
Izod Impact, ft.lbs./in.	0.81	1.70

* Nominal 0.953 g/cc density and 5 g/10 min. MI



LyondellBasell Industries
P.O. Box 3646
Houston, TX 77252-3646
United States

www.LYB.com

Before using a product sold by a company of the LyondellBasell family of companies, users should make their own independent determination that the product is suitable for the intended use and can be used safely and legally. SELLER MAKES NO WARRANTY; EXPRESS OR IMPLIED (INCLUDING ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE OR ANY WARRANTY) OTHER THAN AS SEPARATELY AGREED TO BY THE PARTIES IN A CONTRACT.

LyondellBasell prohibits or restricts the use of its products in certain applications. For further information on restrictions or prohibitions of use, please contact a LyondellBasell representative.

Users should review the applicable Safety Data Sheet before handling the product.

Adflex, Adstif, Adsyl, Akoafloor, Akoalit, Alathon, Alkylate, Amazing Chemistry, Aquamarine, Aquathene, Arcopure, Arctic Plus, Arctic Shield, Avant, Catalloy, Clyrell, CRP, Crystex, Dexflex, Duopac, Duoprime, Explore & Experiment, Filmex, Flexathene, Glacido, Hifax, Hiflex, Histif, Hostacom, Hostalen, Ideal, Integrate, Koattro, LIPP, Lucalen, Luflexen, Lupolen, Lupolex, Luposim, Lupostress, Lupotech, Metocene, Microthene, Moplen, MPDIOL, Nerolex, Nexprene, Petrothene, Plexar, Polymeg, Pristene, Prodflex, Pro-Fax, Punctilious, Purell, SAA100, SAA101, Sequel, Softell, Spherilene, Spheripol, Spherizone, Starflex, Stretchene, Superflex, TBAc, Tebol, T-Hydro, Toppyl, Trans4m, Tufflo, Ultrathene, Vacido and Valtec are trademarks owned or used by the LyondellBasell family of companies.

Adsyl, Akoafloor, Akoalit, Alathon, Aquamarine, Arcopure, Arctic Plus, Arctic Shield, Avant, CRP, Crystex, Dexflex, Duopac, Duoprime, Explore & Experiment, Filmex, Flexathene, Hifax, Hostacom, Hostalen, Ideal, Integrate, Koattro, Lucalen, Lupolen, Metocene, Microthene, Moplen, MPDIOL, Nexprene, Petrothene, Plexar, Polymeg, Pristene, Pro-Fax, Punctilious, Purell, Sequel, Softell, Spheripol, Spherizone, Starflex, Tebol, T-Hydro, Toppyl, Tufflo and Ultrathene are registered in the U.S. Patent and Trademark Office.