INFUSE™ Olefin Block Copolymers

Injection Molding Processing Guide

Introduction

INFUSE™ Olefin Block Copolymers (OBCs) are a ground-breaking family of resins that offer performance previously thought to be unattainable with olefin-based elastomers. Invented using INSITE™ Technology – Dow's approach to linking the science of catalyst, process, and materials together to address customer needs – these resins feature novel block architecture that enables an advantaged balance of flexibility and heat resistance compared to random polyolefin copolymers. The result is an exciting alternative to a broad range of traditional flexible materials.

The OBC product offering features resins for a variety of fabrication processes – creating a world of possibilities for new products, new applications, and new markets. This guide focuses on the use of OBCs in the injection molding process.

OBC Structure and Performance

Polymer chains for traditional block copolymers, such as styrene-butadiene-styrene (SBS), have nearly constant block lengths due to the use of living anionic polymerization. In contrast, Dow's use of catalytic shuttling technology in a continuous solution polyethylene process generates variable, yet controllable, distribution of block length and number. This distinctive process means that INFUSE OBC technology can offer an unprecedented capability to formulate tailor-made olefin polymers for specific end-use applications. OBCs are further characterized by a broader molecular weight distribution (MWD) compared to traditional anionic block copolymers and have the processability of current polyethylene products.

INFUSE OBC resins consist of chains with alternating blocks of "hard" (highly rigid) and "soft" (highly elastomeric) segments. This distinct combination makes them an exciting alternative to styrenic block copolymers (SBCs), ethylene vinyl acetate (EVA), flexible polyvinyl chloride (f-PVC), thermoplastic vulcanizates (TPVs), and more. When compared head to head, INFUSE OBCs enable:

- Improved temperature performance, better weatherability, and lighter weight versus SBS
- Higher heat resistance, lighter weight, and improved compression set versus EVAs
- Lighter weight and enhanced temperature performance versus f-PVC, while offering the sustainability attributes of olefins

All of this translates to an incredible range of opportunities in flexible molded goods, flexible gaskets and profiles, foams, elastic films, adhesives, and many other applications.

OBC Products for Injection Molding

OBC grades designed for injection molding applications are available in a range of densities with melt indices of 5 and 15 g/10 min.

INFUSE™ OBCs can be blended at the press with materials such as HDPE, PP, $CaCO_3$ concentrates, and oil concentrates. Blending OBCs with products such as HDPE or PP offers opportunities to specifically target a desired set of physical properties. OBCs can also be used as a compatibilizer for blending HDPE and PP together. OBC blends with $CaCO_3$ and oil concentrates offer the flexibility of "dialing in" a desired set of haptic properties. The addition of $CaCO_3$ can also enable shorter cycle times when compared to neat OBCs.

For additional information on blending INFUSE OBCs with other materials, contact your Dow sales or technical service and development (TS&D) representative.

General Equipment Considerations

INFUSE™ Olefin Block Copolymers can typically be fabricated on conventional injection molding equipment using the established techniques practiced in the plastics industry. Regardless of the equipment used, the general guidelines provided in the following sections may be helpful in obtaining optimum results.

Several factors must be considered when fabricating INFUSE OBCs into finished goods by injection molding, these include machine selection, material and mold temperatures, screw speed, hydraulic pressure, back pressure, injection speed, and melt cushion.

Machine Selection

Clamp Tonnage

The actual required tonnage depends on a number of factors, including part design, polymer viscosity, polymer flow length, mating mold surface condition, and mold construction. The mold should occupy approximately 2/3 of the platen area between the tie bars to prevent possible damage to the mold from "platen wrap-around." Using the minimum clamp force required to produce acceptable parts will reduce the center deflection of the mold.

Barrel Capacity

The size of the barrel to be used is determined by the volume of polymer required to fill the mold cavity. It is generally recommended that the shot size be kept between 50 and 85 percent of the total barrel capacity. A shot size at the upper end of this range will reduce the material residence time at processing temperatures and minimize the risk of thermal degradation. Exceeding the recommended range may result in a non-homogeneous melt.

Screw Selection

Three general guidelines for selecting an injection molding machine screw for INFUSE OBCs include:

1. A minimum of 20:1 length-to-diameter (L/D) ratio
2. A compression ratio between 2.5:1 and 3.5:1
3. A general purpose polyolefin screw and a general purpose nozzle

NOTE: When using color, $CaCO_3$ concentrates, or making blends at the press, pay particular attention to screw design. Certain high performance or mixing screws have been shown to dramatically improve melt homogeneity when compared with general purpose screws.

Processing Parameters

Barrel Temperatures

Range: 370 to 390°F (190 to 200°C)

The optimum temperature profile depends on many variables, including machine capacity to shot size ratio, screw design, mold and part design, and cycle time. Generally, barrel temperature controllers should be of PID type, and set so the material melts gradually, with cooler rear zone and hotter front zone temperatures. For vented barrel machines, a relatively flat temperature profile is recommended to ensure the polymer is melted by the time it reaches the vent zone. Reverse temperature profiles are used occasionally to compensate for improper screw design, to reduce machine amperage or torque requirements, and to compensate for machines with short L/D ratios. INFUSE OBCs offer a wide processing temperature window for injection molding operations. Suggested melt temperatures for OBCs are summarized in Table 1 (page 3). These temperatures should be used as a starting point and can be increased by a maximum of +35°F (20°C).

Drying

Drying of INFUSE OBCs is not required.

Mold Temperature

Range: 45 to 60°F (7 to 16°C)

Generally, cooling time is the rate-determining factor for overall cycle time. To obtain the best part properties and consistent dimensional tolerances, uniform heat removal is critical. Using a mold temperature controller will minimize temperature variations. Recommended mold temperatures range from 45 to 60°F (7 to 16°C), with cooling times between 20 and 60 seconds, depending on the INFUSE™ OBC resin chosen.

Cooling lines should be properly placed and spaced around the part for effective heat removal. The cooling lines should be adequately sized, without restrictions in the connectors or associated piping. The flow rate of the cooling medium should be sufficient to provide for turbulent flow through the cooling lines. Cleanliness of the cooling medium should also be maintained to prevent blockage of the cooling lines.

Table 1: Suggested Processing Temperatures

Material	Nozzle, °F (°C)	Front, °F (°C)	Center, °F (°C)	Rear, °F (°C)
OBC, ≥ 5 g/10min Melt Index	370 (190)	370 (190)	370 (190)	250 (120)
OBC – HDPE Blends	370 (190)	370 (190)	370 (190)	250 (120)
OBC – PP Blends	390 (200)	390 (200)	390 (200)	250 (120)
OBC – $CaCO_3$ Concentrate Blends	370 (190)	370 (190)	370 (190)	250 (120)
OBC – Oil Concentrate Blends	385 (196)	385 (196)	385 (196)	250 (120)

Screw Speed

For optimum homogeneity of the polymer melt, slow screw speeds are generally suggested. Typical screw speeds range from 40 to 140 rpm. Screw rotation should occur throughout the cooling cycle. This situation can best be accomplished by establishing a proper balance between screw speed and back pressure – that is, a ratio of speed and pressure that will allow plasticization to continue throughout the entire cooling cycle.

Hydraulic Pressure

The injection molding process is generally divided into two stages. The first is injection (or fill), and the second is packing (or hold). During the first – or filling – stage, it is suggested that you set your machine pressure near its maximum setting and control the speed of the ram with velocity controls. The machine will only use whatever pressure is necessary to move the ram at the set speed. This technique will help produce consistent parts because the cavity is filled at a uniform rate, despite differences in viscosity due to temperature fluctuations and other factors.

It is very important to switch from the first to second stage when the part is 90 to 95 percent full. If you do not switch from first to second stage pressure before the cavity is full, the high pressure of the first stage pump may cause the part to flash and may cause damage to the mold. It is suggested to switch from first to second stage based on either the position of the ram or the cavity pressure. Switch-over based on the hydraulic pressure and time is suggested if ram position or cavity pressure controls are not available. While maintaining a cushion, your second stage/packing pressure should be set at the lowest necessary level to properly pack out the part to achieve the desired surface aesthetics. In general, packing pressure is lower than filling pressure. Under-packing the part may result in excessive sink marks, poor dimensional tolerances, and poor surface aesthetics. Over-packing may result in excessive molded-in stress, increased part weight, and poor part performance. Pack/hold pressures of 50 to 80 percent of injection pressures are suggested.

Back Pressure

Suggested screw back pressures range from 50 to 150 psi. Back pressure is necessary to ensure uniformity of the polymer melt for maximum part performance. It is particularly important to use sufficient back pressure when coloring in-house with color concentrates. Operating at the upper end of the suggested range may be necessary to achieve good distributive mixing and uniform part color.

Injection Speed

Injection speed depends on the particular part and machine. Since erratic injection speed can cause a variety of part defects, a uniform injection speed is best. An injection speed of 0.2 to 2.0 seconds is suggested for INFUSE™ OBCs. High injection speeds will create high shear as the melt passes through the runners, gates, and along the cavity surface. OBCs are shear dependent, increasing the shear rate will decrease the polymer viscosity, which allows the material to flow easier and helps fill thin-wall parts. However, excessive injection speed may result in flash, thermal degradation due to shear heating, or burning due to inadequate venting.

Melt Cushion

The use of a "cushion" or residual polymer melt in the barrel after injection will help ensure that the proper amount of material is injected into the cavity. During the packing phase, the material in the cavity is shrinking. To compensate for this shrinkage, additional material must be supplied to the cavity until gate freeze-off.

A small melt cushion provides a ready source of additional polymer to use during packing. If the screw is allowed to "bottom-out," the packing pressure cannot be transferred through the polymer to pack out the cavity. This may result in poor part consistency due to short shots, poor dimensional stability, excessive sink marks, and/or poor aesthetics.

It is generally suggested that a small cushion size be employed, 3 to 6 mm for INFUSE™ OBCs. This will help minimize the residence time and heat history of the polymer, reducing the potential for polymer degradation.

Purging

Clean the hopper and feed throat of the machine carefully when switching materials to avoid contamination. During machine shutdown conditions, material should not remain in the barrel for long periods of time. In preparation for a shutdown, purge the machine with low density polyethylene (LDPE) or general purpose polystyrene (GPPS) before shutdown.

General Rules for Effective Troubleshooting

INFUSE OBCs can typically be injection molded into quality parts with excellent aesthetics. While molding problems do occasionally occur, most can be readily resolved or prevented with attention to the following recommendations.

Understand the Process and Product

A thorough understanding of both the theory and mechanics of the injection molding process, as well as the machinery and other equipment employed, is essential to trouble-free operation. This also applies to the particular resins being processed. Operators should be reasonably familiar with the chemical, physical, and mechanical properties of these materials and should be able to anticipate the probable behavior of the resin under varying heat and pressure conditions. Anticipating potential problems and correcting them before they occur can significantly shorten start-up and production times.

Have a Plan

Plan your approach before taking action. Decide in advance which controls or other factors should be changed and in which order. Remember, troubleshooting without a plan is likely to create rather than solve problems.

Change Only One Setting or Condition at a Time

This change may seem obvious, but experience suggests that this is the one rule that is most often neglected. Clearly, if more than one setting is changed at a time, there is no way to monitor which setting was responsible for the change in machine operation.

Allow Sufficient Time for a Change to Take Effect

Allow sufficient time (e.g., 10 shots) after each change for the material and the machine to adjust and reach equilibrium. If sufficient time is not allowed before additional changes are made, there will be no way to accurately monitor whether or not a particular change was effective. "Sufficient time" depends on the nature of the setting or other factors that were changed, such as the shot size (especially in relation to the barrel capacity), and the length or complexity of the overall cycle.

Keep an Accurate Record or Log of Each Change

Keeping an accurate and detailed log of each change and saving it as a permanent record of the total job is an excellent practice. This record should include the machine number, the material or resin lot numbers, machine conditions and performance, mold and part identification numbers, etc. Such a record can be very helpful the next time the same machine, mold, or material is used. This record can also be extremely useful in preparing future cost estimates for molded parts.

Provide Clear, Written Instructions

When a solution to a problem has been found, post clear, written instructions on or near the machine in question. Also enter these instructions in the log or job history.

Injection Molding Troubleshooting Guide

The following troubleshooting guide lists possible causes and corrective actions for problems typically encountered in the injection molding process. However, this information is not a complete guide to all injection molding challenges and problems. For additional troubleshooting information, contact your Dow TS&D representative.

Splay and Streaks

Splay and streaks (i.e., small voids just below the surface of a molded part) typically are caused by gasses, moisture, or volatiles which have been trapped in the melt during molding. This defect may also be caused by the presence of small amounts of incompatible contaminants.

Possible Cause	Corrective Action
Contamination	• Check for the presence of contamination in the polymer granules • Clean the hopper, air lines, etc. • Thoroughly purge the machine
Excessive shear	• Reduce injection speed • Increase number of vents • Design of part radii • Increase gate size
Excessive polymer temperature	• Reduce injection speed • Reduce back pressure • Reduce screw speed • Reduce barrel temperature set points • Increase gate size
Air trapped in the polymer in the barrel	• Use a graduated temperature profile in the barrel, with the coolest area in the feed zone • Use as little decompression as possible • Increase back pressure

Sink Marks

Sink marks or depressions usually form over the thicker sections of a part. This is due largely to the difference in cooling rates between the thick and thin sections. As thicker sections cool, the molten polymer contracts or shrinks, pulling the surface skin toward the center. The result is usually the formation of a depression or sink mark on the surface.

Possible Cause	Corrective Action
Insufficient polymer in the mold to allow for shrinkage	• Increase packing pressure • Increase injection hold time • Increase amount of feed and verify melt cushion • Increase injection speed • Increase gate size • Check for balanced flow in multi-cavity tools • Check mold cooling • Redesign part to minimize or eliminate thick sections
Polymer "flowing back" out of the mold into the runner system and barrel	• Increase injection hold time. Generally, gates must be sealed before hold time has expired and pressure is removed • Decrease mold temperature and/or increase cooling time
Polymer temperature too high	• Reduce back pressure • Reduce barrel temperature • Improve mold temperature controls
Ejected parts too hot	• Check mold cooling system • Increase cooling time

Weld Lines

Weld lines form whenever two molten polymer flow fronts meet. Problems in strength and appearance occur when there is insufficient knitting or joining of the flow fronts. This problem can be largely corrected by adjusting certain processing conditions. However, since such changes can also affect gloss levels, it is often necessary to strike a balance between gloss levels and the visibility of weld lines. Changing the location of the gates may also be used to move weld lines to "non-critical" locations.

Possible Cause	Corrective Action
Slow injection speed	• Increase injection speed
Under-packing of the part	• Increase hold pressure
Polymer too cold	• Increase back pressure • Increase melt temperature • Increase mold temperature • Increase injection speed
Trapped air or the inability of air to escape from the mold quickly enough	• Clean the vents • Provide additional vents
Multiple gates	• Decrease the number of gates • Increase gate size • Change gate locations

Poor Color Distributive Mixing

Variations in color, including color streaks and swirls, are generally caused by inadequate distribution or mixing of the color concentrate in the natural resin.

Possible Cause	Corrective Action
Inadequate blending of the color concentrate and the natural resin	• Verify the correct let-down ratio • Verify the uniformity of the blend • Install in-line mixers
Inadequate mixing in the barrel	• Increase back pressure • Vary or adjust screw speed • Increase screw return time • Increase barrel temperature • Run a reverse temperature profile in the barrel, with the hottest areas in the feed zone and the coolest in the front zone • Install a mixing nozzle • Install a mixing screw

Burning/Discoloration

Burn marks and/or discoloration are common causes of part rejection. These types of defects are generally caused by trapped gasses in the cavity or thermal degradation of the polymer. Air trapped in the cavity by an emerging flow front will become compressed and superheated. This will result in burn marks on the polymer. Thermal degradation occurs when the material is processed at excessive temperatures or is held at processing temperatures for extended periods of time. These problems can usually be corrected with changes to processing conditions or mold design. Parts that exhibit burn marks or thermal degradation should be discarded and not used as regrind material.

Possible Cause	Corrective Action
Trapped gasses in the cavity	• Clean vents • Decrease injection speed • Provide additional vents
Thermal degradation	• Decrease injection speed • Decrease melt temperature • Determine proper heater band operation • Increase gate size • Check for contamination • Decrease back pressure

Parts Sticking

Parts sticking to the tool can be the result of a variety of factors. Modifications to the processing conditions may help to reduce or eliminate the problem, but in many cases, modifications to the tool may be required.

Possible Cause	Corrective Action
Over-packing of the part	• Reduce holding pressure • Reduce injection speed • Reduce shot size
Insufficient cooling	• Ensure proper operation of the mold temperature controller • Check for turbulent flow of the cooling medium • Verify adequate cooling, and install additional lines if necessary
Material temperature too high	• Decrease back pressure • Decrease injection speed • Decrease melt temperature
Undercuts/Tool finish	• Eliminate undercuts • Polish the tool, especially projections, in the direction of draw
Insufficient draft	• Add additional draft to the part
Mold temperature	• Increase core temperature • Decrease cavity temperature
Excessive cooling time	• Reduce cooling time

For more information on products, innovations, expertise, and other services available to you from Dow Elastomers, visit www.dowelastomers.com or contact us as indicated below.

North America	
U.S. & Canada	1-800-441-4369
	1-989-832-1426
Mexico	+1-800-441-4369

Latin America	
Argentina	+54-11-4319-0100
Brazil	+55-11-5188-9000
Colombia	+57-1-219-6000
Mexico	+52-55-5201-4700

Europe/Middle East	+800-3694-6367
	+32-3-450-2240
Italy	+800-783-825

South Africa	+800-99-5078

Asia Pacific	+800-7776-7776
	+603-7965-5392

The principles of Responsible Care® and Sustainable Development influence the production of printed literature for The Dow Chemical Company ("Dow"). As a contribution towards the protection of our environment, Dow's printed literature is produced in small quantities and on paper containing recovered/post-consumer fiber and using 100 percent soy-based ink whenever possible.

NOTICE: Any photographs of end-use applications in this document represent potential end-use applications but do not necessarily represent current commercial applications, nor do they represent an endorsement by Dow of the actual products. Further, these photographs are for illustration purposes only and do not reflect either an endorsement or sponsorship of any other manufacturer for a specific potential end-use product or application, or for Dow, or for specific products manufactured by Dow.

NOTICE: No freedom from infringement of any patent owned by Dow or others is to be inferred. Because use conditions and applicable laws may differ from one location to another and may change with time, the Customer is responsible for determining whether products and the information in this document are appropriate for the Customer's use and for ensuring that the Customer's workplace and disposal practices are in compliance with applicable laws and other governmental enactments. Dow assumes no obligation or liability for the information in this document. **NO WARRANTIES ARE GIVEN; ALL IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE ARE EXPRESSLY EXCLUDED.**

NOTICE: If products are described as "experimental" or "developmental": (1) product specifications may not be fully determined; (2) analysis of hazards and caution in handling and use are required; (3) there is greater potential for Dow to change specifications and/or discontinue production; and (4) although Dow may from time to time provide samples of such products, Dow is not obligated to supply or otherwise commercialize such products for any use or application whatsoever.

NOTICE REGARDING MEDICAL APPLICATION RESTRICTIONS: Dow will not knowingly sell or sample any product or service ("Product") into any commercial or developmental application that is intended for:
a. long-term or permanent contact with internal bodily fluids or tissues. "Long-term" is contact which exceeds 72 continuous hours;
b. use in cardiac prosthetic devices regardless of the length of time involved ("cardiac prosthetic devices" include, but are not limited to, pacemaker leads and devices, artificial hearts, heart valves, intra-aortic balloons and control systems, and ventricular bypass-assisted devices);
c. use as a critical component in medical devices that support or sustain human life; or
d. use specifically by pregnant women or in applications designed specifically to promote or interfere with human reproduction.

Dow requests that customers considering use of Dow products in medical applications notify Dow so that appropriate assessments may be conducted.

Dow does not endorse or claim suitability of its products for specific medical applications. It is the responsibility of the medical device or pharmaceutical manufacturer to determine that the Dow product is safe, lawful, and technically suitable for the intended use. **DOW MAKES NO WARRANTIES, EXPRESS OR IMPLIED, CONCERNING THE SUITABILITY OF ANY DOW PRODUCT FOR USE IN MEDICAL APPLICATIONS.**

This document is intended for use within North America
Published March, 2009.

Form No. 788-09201-0309 SMG
SMG 11697