

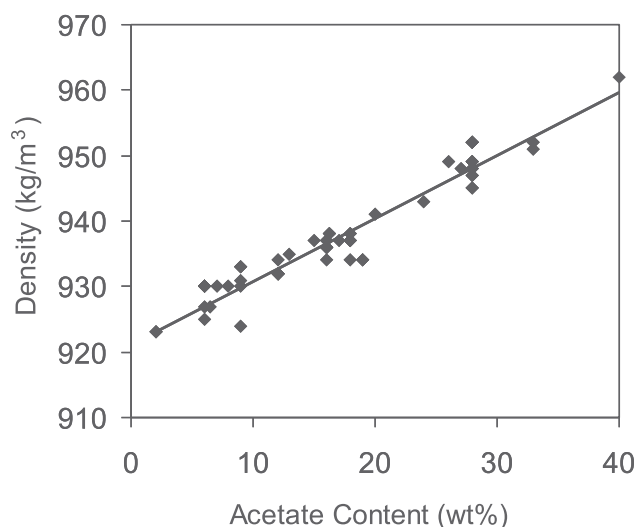
INJECTION MOLDING GUIDE FOR LDPE AND EVA COPOLYMERS

INTRODUCTION

Low-density polyethylene is produced through the polymerization of ethylene in a high-pressure process. By copolymerizing ethylene (E) with vinyl acetate (VA), Ateva® EVA copolymers are produced to form significantly different materials. As the VA content increases, the flexibility, resilience and transparency increase, and the Shore A hardness decreases. However, the softening point

(e.g. Vicat) will also lower. Increasing vinyl acetate content decreases crystallinity and influences all properties related to crystallinity. However, the density increases, due to the higher molecular weight of acetate over ethylene. Note that this is opposite to polyethylene, in which a decrease in crystallinity correlates to a decrease in density.

Figure 1. Correlation between acetate content and density.

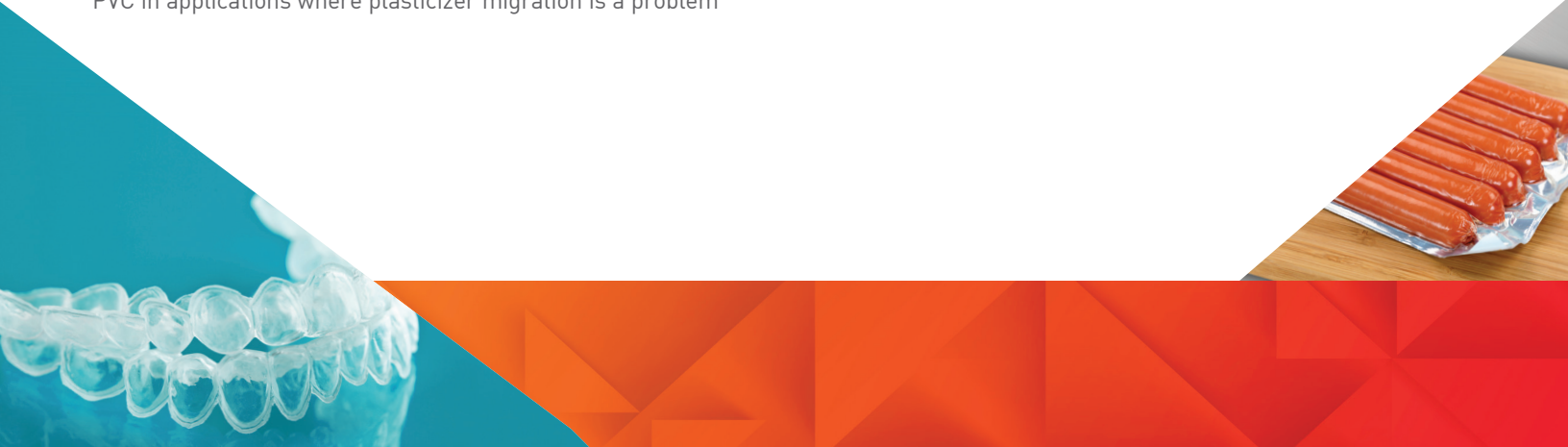


AS ACETATE CONTENT INCREASES:

VICAT SOFTENING POINT	DECREASES
DSC MELTING POINT	DECREASES
TENSILE STRENGTH	DECREASES
TENSILE ELONGATION	INCREASES
FLEXURAL MODULUS	DECREASES
LOW TEMPERATURE BRITTLINESS	IMPROVES
ENVIRONMENTAL STRESS CRACKING	IMPROVES
CHEMICAL RESISTANCE	DECREASES
COMPATIBILITY WITH FILLERS	IMPROVES

This combination of properties makes EVA copolymers suitable as replacements for rubbers and plasticized PVC in a wide range of applications. EVA copolymers can be used to replace PVC in applications where plasticizer migration is a problem

or where low-temperature flexibility is important. The inherent flexibility and fast processing characteristics of EVA make it an attractive alternative to natural and synthetic rubbers.



Typical grades recommended for injection molding are shown in the following two tables. Please consult our website at <http://www.celanese.com/eva-polymers/products.aspx> for a complete listing of grades and information sheets.

Table 1. Ateva® EVA copolymers suitable for injection molding.

Grade	Acetate Content (wt%)	Melt Index (g/10min)	DSC Melting Point (°C)	Vicat Softening Point (°C)	Flexural Modulus (1% Secant)	Shore A Hardness
Ateva 1030	7	1.5	103	86	136	96
Ateva 1070	9	2.8	101	81	101	96
Ateva 1075A	9	8.0	98	79	115	94
Ateva 1231	12	3.0	97	75	82	95
Ateva 1608	16	8.4	90	63	-	92
Ateva 1820	18	3.0	87	-	52	90
Ateva 2020	20	20	83	51	-	89
Ateva 2604A	26	4.0	76	46	-	84
Ateva 2810A	28	6.0	73	44	19	79

Table 2. Celanese LDPE grades suitable for injection molding.

Grade	Melt Index (g/10min)	Density (kg/m ³)	DSC Melting Point (°C)	Vicat Softening Point (°C)	Flexural Modulus (1% Secant)	Shore A Hardness
AT 220	2.0	921	112	94	215	96
AT 472	6.6	917	-	86	230	94
AT 418	12.0	916	-	89	202	94
AT 280	20	917	105	87	190	94

MOLD DESIGN

The elastomeric behaviour of EVA copolymers makes it possible to mold fairly severe undercuts. Threaded components can often be successfully stripped from the mold, although undercuts should have rounded edges to prevent damage, such as thread smearing.

The elastomeric behaviour of EVA copolymers should be considered when designing ejection systems. Ejector pins, particularly on thin sections, should be of generous cross-section to prevent punch-through. Ejector plates or rings are preferred where practicable. When producing tubular or cylindrical moldings a taper, of about 2°, will aid ejection.

GATES AND RUNNERS

In general most types of gates used for low-density polyethylene are suitable for EVA. Unrestricted and streamlined mold gating is recommended wherever possible. Gates should be located in positions where the flow pattern to all parts of the mold will be uniform as possible. Multiple gating may be used.

Runner systems should be at least 6 mm full round, or the equivalent area in a trapezoidal runner. Hot runner systems, although involving high initial cost, can produce a cost saving where large numbers of moldings are required, but temperatures must be controlled accurately.

VENTING

Mold venting may be necessary to prevent air entrapment. Vents should be located at the furthest points from the gate and, to prevent flashing, should be no deeper than 0.0125 mm (0.0005 in) for the first 6.5 mm (0.25 in) from the mold cavity, after which they should be tapered to allow unrestricted escape of air. To remain effective, vents should be located on the moving part of the tool and should be kept clean.

MOLD SHRINKAGE

This term is generally used for the contraction of the molded part after ejection and cooling. It is expressed as a percentage of the mold dimensions.

The mold shrinkage for EVAs is normally between 1.0% and 2.5%, and is mainly affected by the following factors:

MELT TEMPERATURE - the higher the melt temperature, the greater the shrinkage.

MOLD TEMPERATURE - the higher the mold temperature, the greater the shrinkage.

INJECTION DWELL TIME AND PRESSURE - shrinkage will be smaller for longer dwell times and higher pressures.

SECTION THICKNESS - the thicker the molded section, the slower the cooling and the greater the shrinkage.

ORIENTATION - shrinkage will be greater in the direction of flow than at right angles to it.

GATING - shrinkage is usually greater when pin gates are used than when sprue gates are used.

POLYMER - shrinkage depends on the crystallinity of the polymer. An increase in VA content gives reduced crystallinity and less shrinkage.

MOLDING CONDITIONS

An upper limit of 230°C should be observed with all grades of Ateva® copolymers, as at higher temperatures decomposition may occur. The recommendations included here are for general referencing and may need adjusting for a particular product and process. However, a rough guide to the processing temperature can be estimated from the melt index and vinyl acetate content as shown in Figure 2.

Injection molding can cover a wide range of melt indexes, generally in the range of 2 to 20 g/10 min (190°C, 2.16kg). While lower melt indexes may need to run hotter to compensate for the increased viscosity, it is often beneficial for EVA copolymers with higher VA content. This is due to the increased tackiness and lower strength, which can cause problems with ejection and product quality.

To offset the increased tackiness of EVA copolymers, the extruder can often be run cooler than for parts produced with LDPE. This is due to a decrease in the melting point as the VA content increases.

Figure 2 gives a general guide to the cylinder temperature for EVA copolymers. Nozzles are typically set 10°C hotter.

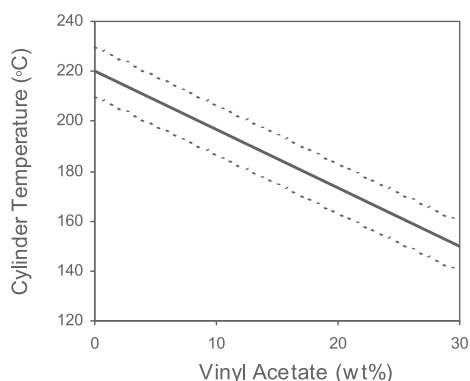


Figure 2. General recommendation for cylinder temperatures.

MOLD TEMPERATURE

To keep cycle times to a minimum and to reduce mold-sticking problems, mold temperatures should be kept below 40°C. However, the use of excessively low temperatures will increase the tendency for 'frozen-in strain' problems to develop. To minimize distortion in the final molding, both mold halves should be at the same temperature.

FROZEN-IN STRAIN

The use of high pressure, high melt temperatures and low mold temperatures in order to reduce cycle times should be avoided if possible, as this is likely to lead to frozen-in strain. Highly strained parts have poor environmental stress cracking resistance and can fail prematurely in service. To produce moldings of consistent good quality, it is important to achieve a correct balance between cycle times, melt temperature and injection pressure.

MOLD RELEASE

When processing the more flexible grades of EVA where ejection can prove troublesome, special attention must be paid to ejection methods and ejector design. With these grades the inclusion of a mold release agent in the polymer can be useful. The addition of 0.1% slip additive and 0.1 - 0.25% antiblock additive provides satisfactory mold release.

Mold release agents applied to the mold surface, such as zinc stearate or PTFE spray, are also useful in preventing mold sticking. Release agents incorporating silicone should be avoided as they can cause stress cracking of the moldings in service.

ADDITIVES

COLORING

EVA copolymers can be colored using conventional masterbatch techniques.

Dry coloring techniques can be used provided the machine produces adequate mixing to disperse the pigment.

UV PROTECTION

EVA copolymers should be protected against UV radiation in outdoor use. 2% of a finely dispersed Carbon Black or 0.05-0.3% of a UV additive should provide satisfactory protection.

CHOICE OF MACHINE

EVA copolymers are easily processed on most molding machines. Although extrusion molding machines and plunger machines can be used, reciprocating screw injection molding machines are preferred as they give a more homogeneous melt, better dispersion of any master batches used, much faster cycle times at lower cylinder pressures and greater flexibility of mold conditions.

Ideally, the plasticizing capacity of the machine should not be greater than twice the shot size. This will minimize the residence time of the material in the cylinder, and will assist in the production of good consistent moldings. With large flat moldings, it may be necessary to disregard this rule as a large machine may be needed to achieve the required locking force.

CHOICE OF MOLD MATERIALS AND FINISHES

Conventional mold steels of the high quality chromium-nickel type should be used for making molds intended for long production runs on EVA. For molds that are likely to be required for very long runs it may be advantageous to produce the mold from a high (12% or greater) chromium steel.

TROUBLESHOOTING GUIDELINES

Defect	Cause	Recommendations
Short shots (Mold not filled)	Insufficient material in cylinder	Verify machine is not pressure limited. Increase pressure if possible. Increase shot size to obtain 95-98% fill
	Blocked nozzle or gate	Clear blocked material from nozzle and/or gate
	Increase in melt viscosity	Verify machine is not pressure limited – increase pressure if possible. Then increase shot size and/or injection speed. Optimize to obtain 95-98% fill at fill-only
	Melt-front freezing off	Verify machine not pressure limited. Then, increase injection speed. If problem remains, increase melt temperature. The most difficult cases may require widening of runners and gates
	Inadequate venting	Verify vents are not shut and that adequate venting is provided at end of fill
Voids and sinks	Unfilled part	Verify that problem is not caused due to short shots
	Underdamped during packing	Verify gate freeze. Increase hold pressure
	Premature gate-freeze	Increase injection speed Increase melt-temperature If problem remains, increase gate size
	Inadequate venting	Verify vents are not shut and that adequate venting is provided at end of fill. Repair tool if necessary
Unmelts	Inadequate plasticization	Increase cylinder temperature
		Lengthen molding cycle and/or slow screw speed
		Consider screw design optimization
		Increase back pressure
Weld lines	Incomplete knitting of divided melt-stream	Weld-lines will occur in areas where divided flow-fronts meet. Best practice is to address during part design to avoid meeting of flow-fronts or locate these sections away from high stress areas
		Weld-lines may be reduced to some extent by:
		Increasing melt temperature
		Increasing mold temperature
Rough surface finish	Low mold temperatures	Increase mold temperature
	Inadequate venting	Check vents and provide adequate venting if necessary
	Creeping flow-front	Increase injection speeds Increase melt temperatures
	Poor mold surface finish	Clean mold surface. Polish if necessary
Flashing at parting lines	Inadequate clamp tonnage	Increase clamp tonnage
	Shot size too large	Verify fill only. Reduce shot size and/or adjust injection speed
	Partially clogged vents	Check and clean all vents to ensure they are open
	Damaged parting line	Repair parting line of tool
Burn marks	Air trapped in mold cavity	Check vents are open and provide adequate venting
		Reduce injection speeds
		Reduce melt temperatures
Warping	Inadequate crystallization (tensile stress) & shrinkage	Increase cooling time Increase mold temperature
	Flow-related (orientation) stress	Use profiled injection speeds during transitions of wall-thicknesses; slow speeds at regions of lower wall thicknesses
	Compressive stress	Ensure fill only to 99%. Reduce hold pressure Ensure part length-to-thickness ratio is not excessive Relocate gate to reduce actual flow length
	Anisotropic shrinkage	Relocate gate to provide symmetric flow paths Add features on tool to provide support Reduce shear by reducing injection speed or increasing melt temperatures
Drag Marks/Part Sticking	Poor mold design	Design core and cavity with small draft
	Excessive friction	Mold release. Either on mold or incorporated in polymer Polish mold surfaces
	Shrinkage	In rare cases, decreases in hold pressure, mold temperatures and cooling time may be used. However, changes in these variables will affect the properties of the final part

CONTACT INFORMATION

Headquarters:

Celanese EVA Polymers
222 W. Las Colinas Blvd., Suite 900N
Irving, TX 75039+

Manufacturing:

Celanese EVA Polymers
4405,101 Avenue NW, Edmonton,
Alberta, Canada T5J 2K1
Phone: (780) 468-0800

Technology and Product Stewardship

Celanese EVA Polymers
8040 Dixie Highway, Florence, KY, USA 41042
Phone: (859) 525-4740
orderseva@celanese.com
celanese.com/eva-polymers

Customer Service: 1-800-661-3663

eva.techservice@celanese.com

Ateva® Product Web Link:
<http://www.celanese.com/eva-polymers/products.aspx>

© 2015 Celanese or its affiliates. All rights reserved.

This publication was printed based on Celanese's present state of knowledge, and Celanese undertakes no obligation to update it. Because conditions of product use are outside Celanese's control, Celanese makes no warranties, express or implied, and assumes no liability in connection with any use of this information. Nothing herein is intended as a license to operate under or a recommendation to infringe any patents.

Celanese®, registered C-ball design and all other trademarks identified herein with ®, TM, SM, unless otherwise noted, are trademarks of Celanese or its affiliates.

EVA-016-PlasticsMoldingGuide-TG-EN-1115