M COM

Alfater XL®
Processing Guide
Injection Molding

Alfater XL® is a thermoplastic vulcanizate elastomer (TPV) based on PP and cross-linked EPDM.

The following series are suggested for injection molding:

- Alfater XL® I 2GP series
- Alfater XL® I 3EF series
- Alfater XL® I 4GP series
- Alfater XL® I 4FC series
- Alfater XL® I 4PA series (separate guide)
- Alfater XL® ECO 2GP series

Processing of TPV is generally different from conventional thermoplastics. The following processing guide aims to provide a first support, especially for customers with less experience in TPV processing.

Pre-drying

Alfater XL® is basically non-hygroscopic. However, pre-drying for 2 - 4 hours at 70 - 80 °C in dry-air (desiccant) dryers is recommended to ensure a stable injection molding process and good surface quality of injection molded parts. Particularly the polyamide-bonding grades, Alfater XL® 4PA, need to be pre-dried before processing. Open bags should not be stored over a long period of time.

Flowability and shrinkage

Harder Alfater XL® grades generally have higher viscosity than softer grades. Due to the cross-linked EPDM phase, the viscosity or flow behavior of Alfater XL® responds much more sensitively to shear than to temperature changes (Figure 1 and Figure 2 on the next page). Appropriate flow, homogeneity, and molding behavior of the Alfater XL® melt is achieved by applying sufficiently high shear. Thus, the injection molding parameters and gating system should be selected and designed correctly.

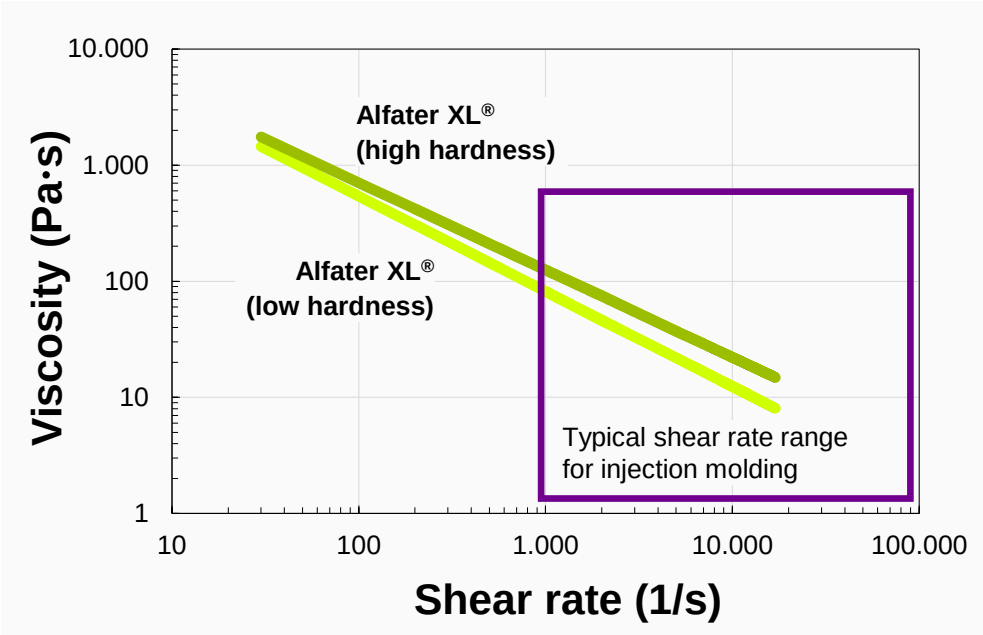


Figure 1: Influence of shearing on the flow behavior (viscosity) of Alfater XL® melt.

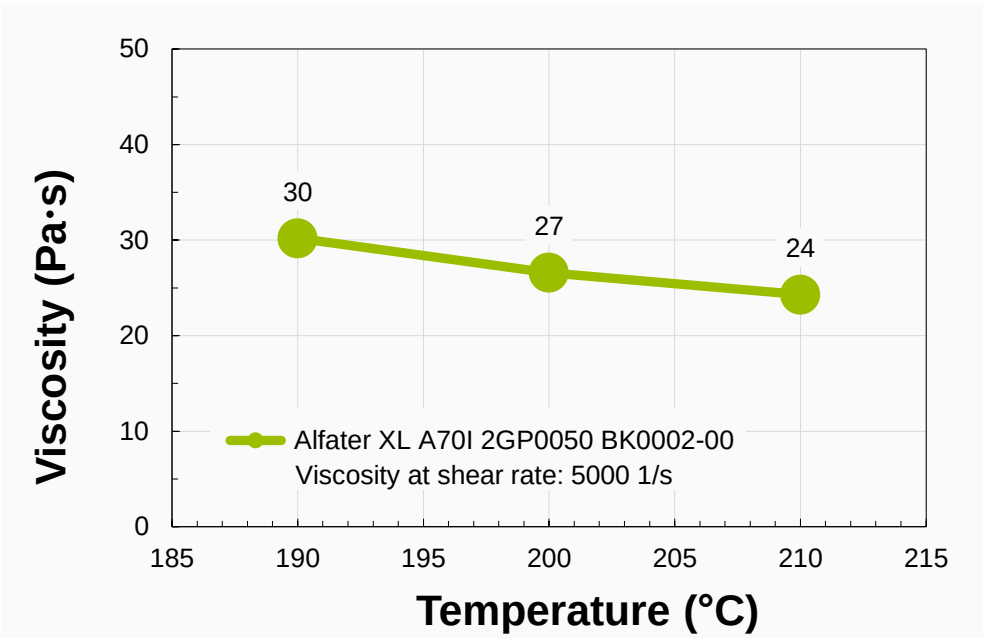


Figure 2: Influence of temperature on the flow behavior (viscosity) of Alfater XL® melt.

Shrinkage of Alfater XL® is anisotropic, which means it shrinks differently in flow direction and transverse to flow direction. Softer Alfater XL® grades tend to shrink more than harder grades. Shrinkage depends strongly on the injection molding conditions and part design. The shrinkage values for the Alfater XL® injection molding series given in the table have been measured on standard sample plates and are therefore only a guideline. Depending on the injection molding process, the processing conditions and the part design, these values can vary.

Hardness of Alfater XL®	Guideline values for shrinkage (ISO 294-4, after 24 h)	
	Parallel to flow	Transverse to flow
< Shore A50	3.0 - 5.0 %	1.5 - 3.0 %
Shore A50 - A90	1.5 - 3.0 %	1.0 - 2.0 %
> Shore A90	1.0 - 2.0 %	0.7 - 1.5 %

In principle, the shrinkage behavior of Alfater XL® can be optimized or minimized to a certain degree by adjusting the injection molding conditions. Important parameters here include the selection of the injection speed, the cooling time and the holding pressure level as well as the holding time. Further information on this topic can be found in our separate shrinkage guideline.

The Alfater XL® injection molding series generally have different flow characteristics. Therefore, not all series are suitable for the same molding part or application. The table below provides a short overview about the general flow behavior and potential use of the Alfater XL® injection molding series.

Alfater XL® series	Flowability	Recommended application
Alfater XL® I 2GP	Normal flow	Standard molding parts, flow length ≤ 150 mm, very good compression set
Alfater XL® I 3EF	Easy flow	Thin-walled parts, complex part designs, flow length > 150 mm, high weathering resistance
Alfater XL® I 4GP	Easy flow	Thin-walled parts, complex part designs, flow length > 150 mm, natural and colored parts, low emission parts
Alfater XL® I 4FC	Easy flow	Thin-walled parts, complex part designs, flow length > 150 mm, parts with food contact
Alfater XL® I 4PA	Easy flow	Thin-walled parts, complex part designs, flow length > 150 mm, 2K parts with polyamide
Alfater XL® ECO 2GP	Normal flow	Standard molding parts, flow length ≤ 150 mm, recyclate and bio-based content

Machinery

Three-zone polyolefin screws are recommended. The L/D ratio of the screw should be ≥ 20:1, preferably 24:1. The compression ratio df/dm of the screw should be between 2:1 and 3:1. Furthermore, the screw design should allow adequate shearing of the Alfater XL®. Specific mixing elements at the screw tip can improve homogeneity of the Alfater XL® melt. Very short and / or low shear screws, such as those used for PVC processing, are not recommended.

The clamping force of the machine should be higher than the separating force caused by the injection process, otherwise quality problems can occur, e.g., flashing.

Injection speed

The viscosity and flowability of Alfater XL® is mainly affected by shear. Therefore, it is recommended to apply high injection speed for Alfater XL® to achieve optimal molding. High injection speed typically

- improves the quality of weld lines
- results in faster mold filling and in improved filling of cavities having long flow paths
- improves quality of the part (e.g., reduction of “tiger stripes”, higher surface gloss, smooth and homogenous part surfaces)
- can lead to air traps (dieselling) if there is insufficient mold venting

Especially for thin-walled parts having long flow paths, it is recommended to use high injection speed. Thick-walled parts and parts with short flow paths can be molded with somewhat lower injection speed. On the other hand, very high injection speed can also cause quality issues depending on part and gate design, e.g., flashing or jetting. Therefore, it is highly recommended to consider the need for high injection speed of TPV in the part and gate design. If required, injection molding simulation data for selected Alfater XL® grades can be provided by the Technical Service Center of MOCOM.

Back pressure and screw speed

Back pressure is generally important for injection molding of Alfater XL®. Too low back pressure will result in insufficient plasticizing and melting of Alfater XL®, which in turn will cause inhomogeneous melt. Sufficient back pressure will provide effective plasticizing and melting of Alfater XL®, good melt homogeneity as well as good dispersion of color masterbatches and additives. However, high back pressure will also lead to longer plasticizing times. The back pressure level to be selected depends, among other things, on the Alfater XL® type and its flowability.

Easy-flow series such as the Alfater XL® I 3EF series basically require lower back pressures. Depending on the screw diameter, a hydraulic back pressure > 3 bar is recommended as a guide value. This means that, depending on the screw diameter, a specific back pressure of > 50 bar should thus be achieved.

Normal flow series such as the Alfater XL® I 2GP series, on the other hand, usually require a higher back pressure for good melt homogeneity. Depending on the screw diameter, a hydraulic back pressure > 5 bar is recommended as a guide value. This means that, depending on the screw diameter, a specific back pressure of > 80 bar should thus be achieved.

The screw speed setting should not be too high in order to avoid excessive shear heating and to achieve good homogeneity of the Alfater XL® melt. As a general rule, a higher screw speed can be used for smaller screw diameters than for large screw diameters. Depending on the screw diameter, the screw speed should be selected so that the peripheral screw speed is preferably 0.2 - 0.3 m/s. In principle, it is more advisable to select a sufficiently high back pressure in combination with a low to medium screw speed.

Holding pressure and holding time

Basically, Alfater XL® has a shear-dependent flowability. For this reason, it is advisable that the mold is almost completely filled during the injection phase - i.e., 98 % or more. Underfilled molds cannot be properly filled in the holding pressure phase. The holding pressure applied, is predominantly for shrinkage compensation. In addition, holding pressure improves weld line quality and adhesion to hard components in 2K molding. The holding pressure level should typically range from 50 - 70 % of the injection pressure. Excessive holding pressure is not advisable because this can lead to part deformations, internal stress or cold flow problems, especially close to the gate.

Holding time should be rather short. Thick-walled parts usually require longer holding times than thin-walled parts. Besides holding pressure, shrinkage can also be compensated to a certain extent by choosing a proper holding time. Longer holding times will reduce shrinkage. It is advisable to generally adjust the holding phase to the gate freeze off time, which can be determined by weighting parts to check when the weight no longer increases with additional holding time.

Temperature profile

A gradually increasing temperature profile is recommended. The temperature profile below is a guideline and a starting point for process optimization.

Temperature adjustments may be necessary depending on the injection molding process applied, the part design, and the Alfater XL® grade used.

Easy flow grades such as Alfater XL® I 3EF, can normally be molded at slightly lower temperatures than normal flow Alfater XL® I 2GP series.

Optimal melt temperature for Alfater XL® should range from 200 - 230 °C. High melt temperature normally

- improves the quality of the part surface
- improves adhesion to polyolefin substrates
- improves the quality of weld lines
- promotes long flow paths and improves mold filling
- results in longer cooling time

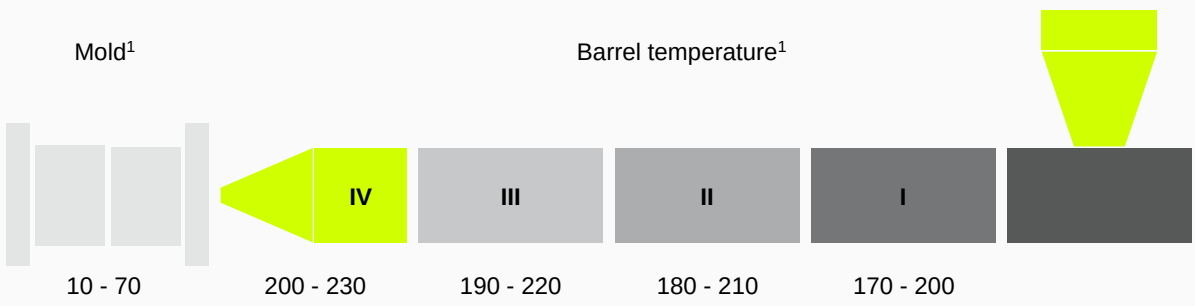
Mass temperatures significantly higher than 230 °C are not recommended, especially in combination with extended residence times in the machine as this can cause thermal degradation of Alfater XL® and / or evaporation of additives.

Mold temperature

Mold temperature should range from 10 - 70 °C depending on part design, injection molding process and hardness of the Alfater XL® grade used. Typically mold temperature between 20 - 50 °C are preferred for most parts and processes. High mold temperatures

- promote long flow paths
- support thin-walled part moldings
- improve the quality of the part surface
- increase cooling time

Mold temperatures above 70 °C are generally not recommended because this can lead to demolding problems, especially in the case of the softer Alfater XL® grades.



Temperatures in degrees Celsius (°C)

[1]Guide values. Standard starting profile might be in the middle.

Gate and mold design

Sprue

The sprue helps to transfer the melt smoothly to the runner system. The sprue should be rather short for Alfater XL®. Sprues with 3° or more are desirable. Tapered or Z-pin sprue-pullers are recommended (Figure 3).

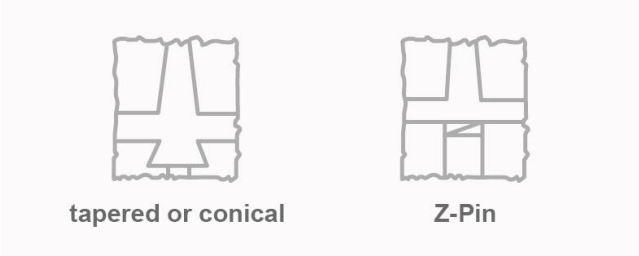


Figure 3: Recommended sprue-pullers for Alfater XL®.

Runner

Today, multi-cavity molds are often used in high volume applications. It is important that all cavities are filled within the same time and with the same pressure. Therefore, the runner should be balanced. If the runner system is not correctly balanced, cavities closer to the sprue will be over-packed while cavities away further from the sprue won't be completely filled (Figure 4).

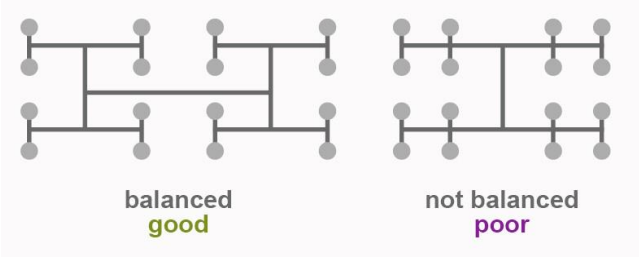


Figure 4: Example for balanced and unbalanced runner design.

If the runners are in one plane, shear edges and dead spaces can occur. This favors asymmetrical mold filling and material degradation. The best solution for optimum mold filling is to use streamlined channels. The runners should be as short as possible and as smooth as possible without dead areas or sharp edges to avoid material degradation. In addition, the dimensions of the channels should be large enough to minimize the pressure drop. Full round runners are preferable, as they provide most efficient cross-section for pressure transmission and heat retention. Trapezoidal runners are also suitable. Half round runners are not recommended (Figure 5).

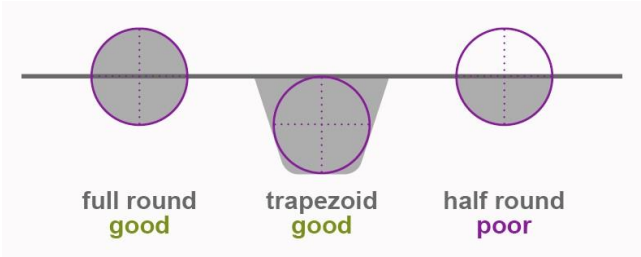


Figure 5: Suitable and unsuitable cross-sections of runner channels for Alfater XL®.

Gate

The gate should be as small as possible, to ensure good melt flow behavior. The shear rate applied via the gate where the Alfater XL® melt enters the cavity should range from 10^4 - 10^5 s^{-1}. The position of the gate should ensure that jetting is avoided, i.e., the incoming melt should make contact with a wall as soon as possible. It is therefore recommended that the gate is able to direct the melt towards the mold wall or pin and not into a free cavity. In addition, the position of the gate should allow the feeding of the thickest section of the part first meaning that material will flow from thick to thin sections.

The length of the gate should be rather short, roughly half of the gate diameter is preferred. Typical gate diameter should range from 0.6 - 1.0 mm. For small parts, the gate diameter could be < 0.6 mm. It is preferred to start with a smaller gate diameter which can easily be increased if required. For large parts (> 150 - 200 mm flow path) or complex part designs it is recommended to use more than one gate for fast and balanced mold filling. It is advisable to conduct injection molding simulations, e.g., Moldflow or Moldex3D for optimal mold design, gate location, and number of gates. Corresponding simulation data are available for selected Alfater XL® types and can be provided on request.

Hot runner

Today, hot runner systems are state-of-the-art in many injection molding processes. Alfater XL® can generally be processed with hot runner systems. The hot runner should be as short as possible to avoid long residence times in the runner. In addition, the runner should be balanced for uniform mold filling. Insulated or internal heated runner molds are not preferred as they promote degraded or unmolten material. Externally heated runners are recommended in conjunction with precise temperature control. Each channel should have its own temperature sensor close to the gate for efficient and precise temperature control. It is also important to ensure homogenous temperature distribution through the complete runner system and to avoid burrs, sharp corners or dead areas as this can promote material degradation. To realize sufficient shearing of the Alfater XL® melt, the hot runner should have rather small channels.

Full round hot runner channels are preferred. The gate for hot runner systems should be small enough for good shear generation. It is advisable that the gating system maintains the melt temperature with minimum heat loss between the hot flow channels and the cooler mold. Open nozzle diameter can be noticeably < 1 mm, for example 0.6 mm. However, open nozzle systems can cause gate vestige, drool and demolding problems due to the elastomeric properties of Alfater XL®. Therefore, open nozzle systems are only recommended for parts having low aesthetic requirements and being used in non-visible applications. Needle valve gates are preferred in any case as they prevent nozzle drool and allow effective shut-off to avoid gate vestige on the molded part. The needle valve diameter should be around 1 mm.

Venting

The high injection speed for processing TPV requires sufficient mold venting. Insufficient venting will cause quality problems such as burn marks, trapped air and voids, mold filling problems as well as poor surfaces and weld line problems. Vents should be placed on the mold parting line, at ejector pins and weld lines as well as at points farthest from the gate to ensure complete mold filling. It is advisable to have separate venting of large runners. The vents should have a depth of ≤ 0.025 mm in the area that is in direct contact with the cavity in order to avoid flash or material escape through the vent. As the distance from the cavity increases, the depth of the vent can also be increased to, for example, 0.2 mm. Figure 6 below shows an example of a possible vent design at the flow path end of the cavity.

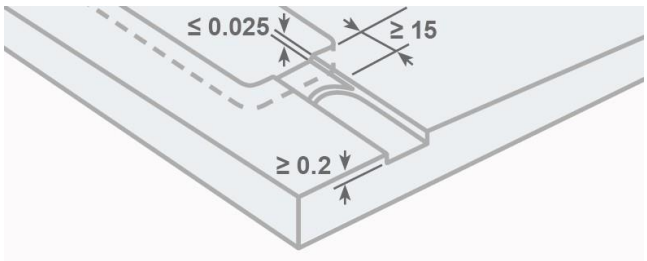


Figure 6: Example for the mold venting design for Alfater XL® (dimensions in mm).

Demolding

Soft Alfater XL® grades generally need more ejectors than hard grades. It is advisable that the surface area (in contact with the part) of the ejector pins are as large as possible to avoid part deformation or piercing.

Forced demolding of parts is possible depending on the part design, undercuts and hardness of Alfater XL®. Basically, pin ejection supported by air ejection is preferred for effective demolding. It is also recommended to distribute the ejectors uniformly around the part in order to avoid asymmetric ejection and distortion of the part.

To ensure proper demolding, draft angles of 1 - 3° are advisable, depending on the part design, the mold surface and the Alfater XL® grade. Softer Alfater XL® grades generally require somewhat larger draft angles than harder grades.

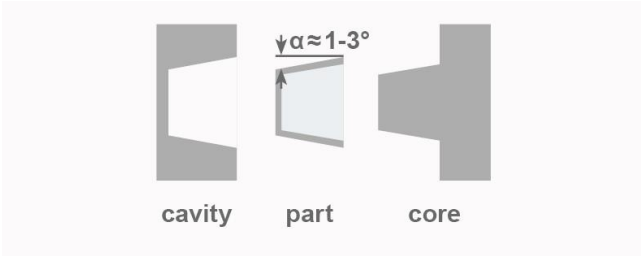


Figure 7: Suggested draft angles for Alfater XL®.

In addition, a matt or even slightly roughened mold surface will facilitate part ejection. A mold roughness of 27 - 33 according to VDI 3400 is useful. In contrast, highly polished mold surfaces are not advisable, especially for soft grades as this merely promotes adhesion of the Alfater XL® to the mold surface. If a polished mold surface is used nevertheless, an adequate mold coating can support or improve the demolding of TPV and reduce the risk of possible deposit formation. The selection of a suitable mold coating should be discussed with the experts from the coating specialist. The Technical Service Center of MOCOM can gladly provide contacts for this purpose.

Cleaning

It is recommended to clean the machine before and after processing of Alfater XL®. PP or PE with a low melt flow rate (e.g., MFR < 5 g/10min) should be used for purging. It is important to remove all PVC and acetal products (e.g., POM) from the machine before processing Alfater XL®. If necessary, the machine has to be cleaned mechanically.

Use of regrind

In principle, production waste from Alfater XL® can be easily recycled and can be used as regrind in a new production in the sense of zero-waste production. The maximum amount of regrind that can be added depends on many factors, e.g., the part design and application. Therefore, the optimal dosage should be defined individually for each process and each part. As a general rule, we would recommend to limit the regrind amount to 20 %. It is essential that the regrind is clean and free from any impurities. Grinding or regranulation of the scrap to a particle size comparable to the granule size of the virgin material is recommended. This can minimize separation problems between the regrind and the virgin material. Pre-drying of the recycled material is also recommended. Pre-drying conditions are similar to prime quality, 2 - 4 hours at 70 - 80 °C in a dry-air (desiccant) dryer.

Coloring

Natural-colored Alfater XL® grades can be easily colored. However, the best choice would be a colored Alfater XL® compound as this ensures the highest color consistency and batch-to-batch repeatability. For requests on a colored Alfater XL®, please contact the Technical Service Center of MOCOM Compounds GmbH & Co. KG or your contact person at our distribution partners.

As an alternative to a directly colored TPV compound, PP-based color masterbatches can also be used. They usually have good compatibility with the Alfater XL®. The color masterbatch used should have sufficient flowability and be suitable for use in injection molding. Typical feeding level of such a batch is 1 - 5 %. However, the optimal dosage can vary depending on the injection molded product or application. It is therefore highly recommended to discuss the best feeding level either with the color masterbatch producer or with your local technical contact at MOCOM or your responsible distribution partner.

Sufficient mixing of the Alfater XL® melt with the color masterbatch is necessary to achieve high quality colored products. Mixing can be improved using appropriate injection molding parameters such as increased back pressure level. In addition, mixing elements such as mixing head or static mixers at the screw tip can further improve the dispersion of the color masterbatch.

MOCOM Compounds GmbH & Co. KG
Mühlenhagen 35 | 20539 Hamburg
T +49 40 78105-720 | sales@mocom.eu
T +49 40 78105-710 | technical@mocom.eu
www.mocom.eu