ALFATER XL®

Injection Molding Grades

PROCESSING GUIDE

ALFATER^XL® is a thermoplastic vulcanizate (TPV) based on PP and cross-linked EPDM. Different ALFATER^XL® series are available and optimized for various injection molding applications:

- **ALFATER^XL® I2GP series**
- **ALFATER^XL® I3EF series**
- **ALFATER^XL® I4GP series**
- **ALFATER^XL® I4FC series**
- **ALFATER^XL® I4PA series**

Typical injection molding applications are shown in Figure 1. Processing of TPV is generally different from conventional thermoplastics. The following injection molding guidelines aim to provide a first support, especially for customers with less experience in TPV processing.

Figure 1: Gaskets produced by injection molding.

Figure 1: Mud flaps produced by injection molding.

DRYING

The ALFATER^XL® injection molding series are basically non-hygroscopic. However, drying for 2 – 4 hours at 70 – 80 °C in a dry-air dryer is recommended to remove any surface moisture. Open bags should not be stored over a long period of time.

FLOWABILITY AND SHRINKAGE

Harder ALFATER^XL® grades have basically higher viscosity than softer grades. The viscosity of ALFATER^XL® is more sensitive to shear than to heat (Figure 2). Appropriate flow, homogeneity, and molding behavior of the ALFATER^XL® melt is achieved by applying sufficiently high shear.

Therefore, appropriate injection molding parameters and gate design are required. In general, the faster the injection speed, the higher the shear rate and the lower the viscosity (= easier flowing). In contrast, temperature increases have only a moderate effect on the flowability and molding behavior of the ALFATER XL® melt.

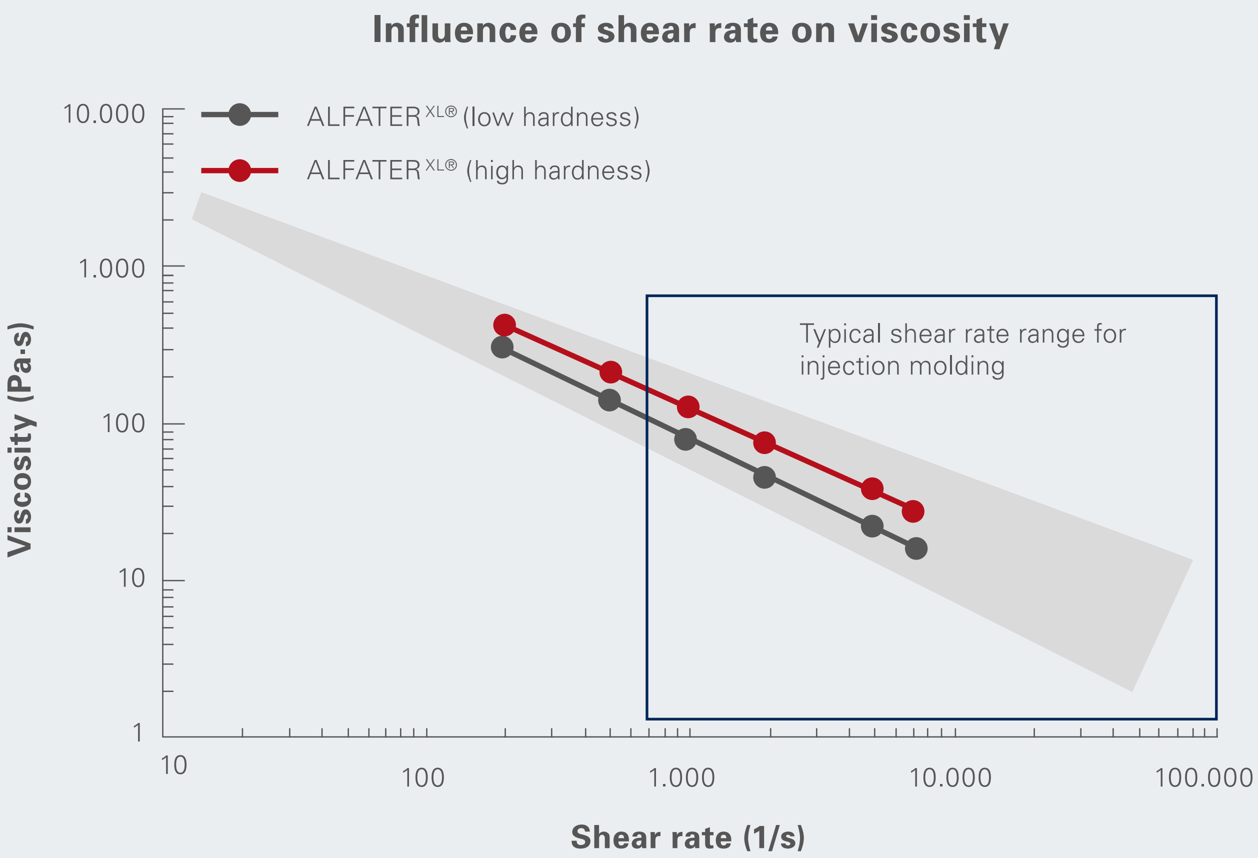
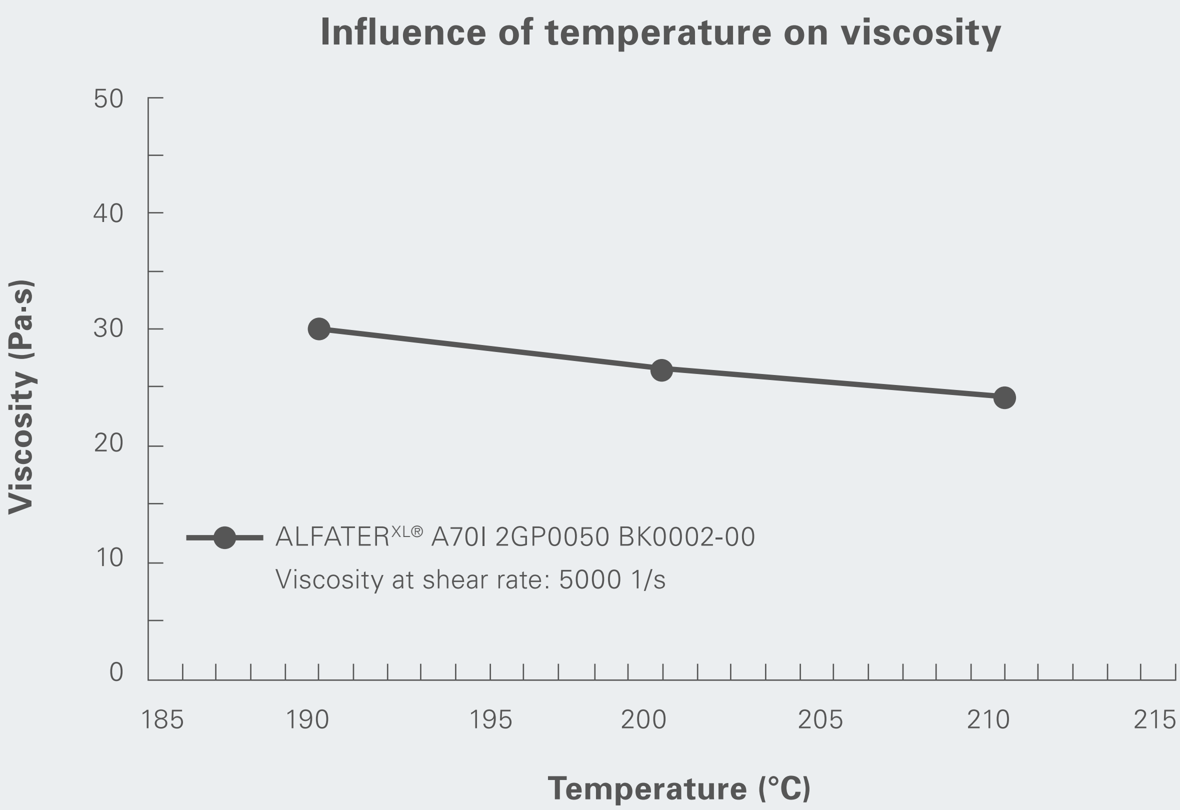


Figure 2: Influence of shearing and temperature on the flow behavior (viscosity) of ALFATER XL® melt.



The ALFATER$^{XL®}$ injection molding series provide different flowability. Therefore not all series are suitable for the same molding part or application. The following table provides a short overview about the general flow behavior and potential use of the ALFATER$^{XL®}$ injection molding series.

Material	Flowability	Spiral flow* (2 mm thickness)	Characteristics
ALFATER$^{XL®}$ I 2GP series	normal flow	< 50 cm	Standard injection molding series, standard molding parts, flow length ≤ 150 mm, best tensile properties and compression set
ALFATER$^{XL®}$ I 3EF series	easy flow	> 50 cm	Thin-walled parts, complex part designs, flow length > 150 mm, excellent weathering resistance
ALFATER$^{XL®}$ I 4GP series	easy flow	> 50 cm	Thin-walled parts, complex part designs, flow length > 150 mm, natural and bright colored parts, low emission parts
ALFATER$^{XL®}$ I 4FC series	easy flow	> 50 cm	Thin-walled parts, complex part designs, flow length > 150 mm, parts for food contact

* Spiral flow: thickness 2 mm, melt temperature 220 °C, mold temperature 70 °C, injection pressure 500 bar

Shrinkage of ALFATER$^{XL®}$ is anisotropic (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 below have been measured on standard sample plaques 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$^{XL®}$	Shrinkage (ISO 294-4), after 24 h	
	Parallel to flow	Transverse to flow
Shore A30 - A50	3.0 – 5.0 %	1.5 – 3.0 %
Shore A50 - A90	1.5 – 3.0 %	1.0 – 2.0 %
Shore A90 - D50	1.0 – 2.0 %	0.7 – 1.5 %

Shrinkage of ALFATER$^{XL®}$ can be optimized to a certain degree by adjusting the injection molding conditions. Important molding parameters and their influence on the shrinkage are listed below in the table.

Injection molding parameter	Change	Shrinkage
Injection speed	↑	↓
Holding pressure / holding time	↑	↓
Mold temperature	↓	↓
Cooling time	↑	↓

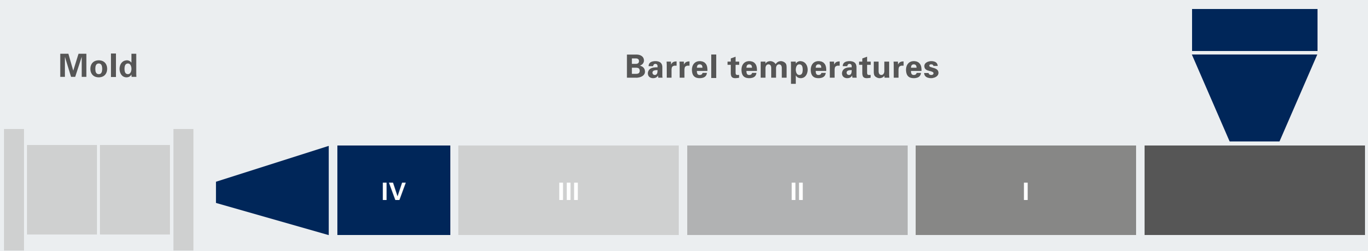
MACHINERY AND EQUIPMENT

A three zone polyolefin screw with a screw length L > 18D and a compression ratio df/dm of 2:1 to 3:1 is recommended. The screw design should allow sufficient shearing of the ALFATER$^{XL®}$. Specific mixing elements at the screw tip can improve homogeneity of the ALFATER$^{XL®}$ melt. Screw designs with low shear input, e.g. screws for PVC processing, are not recommended.

The clamping force of the machine should be higher than the separating force caused by the injection process of molten ALFATER$^{XL®}$ into the mold. Quality problems can occur, e.g. flashing, if the clamping force is insufficient.

TEMPERATURE PROFILE

A gradually increasing temperature profile is recommended. The temperature profile below is a guideline and a starting point for process optimizations (Figure 3). Temperature adjustments may be necessary depending on the injection molding process applied, the part design, and the ALFATER$^{XL®}$ grade used. Easy flow grades, e.g. ALFATER$^{XL®}$ I 3EF, can normally be molded at sligthly lower temperatures than normal flow ALFATER$^{XL®}$ I 2GP series.



Temperatures in degrees Celsius (°C)[1]

Mold	IV	III	II	I
10 – 70	200 – 230	190 – 220	180 – 210	170 – 200

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

Figure 3: Recommended temperature profile for injection molding of ALFATER$^{XL®}$
(the temperatures are only a guideline).

Optimal temperature of the ALFATER$^{XL®}$ melt 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

Temperatures significantly higher than 260 °C are not recommended, especially in conjunction with long residence times e.g. in hot runner systems, because 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. Typically mold temperature between 20 – 50 °C are suitable for most parts and processes. High mold temperatures:

- promotes long flow paths
- supports thin-walled part moldings
- improves the quality of the part surface
- increases cooling time

Mold temperatures higher than 70 °C are not recommended, because this can result in demolding problems of ALFATER$^{XL®}$.

INJECTION SPEED

The viscosity and flowability of ALFATER$^{XL®}$ is significantly 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
- promotes long flow paths
- results in faster and improved mold filling
- 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. ALBIS PLASTIC GmbH can support by providing ALFATER$^{XL®}$ data for injection molding simulations, such as Moldflow and Moldex3D.

BACK PRESSURE AND SCREW SPEED

The hydraulic back pressure should be ca. 5 bar or higher. Depending on screw diameter this should result in a back pressure (specific) of at least 50 bar or higher. Easy flow grades basically require lower back pressure level than normal flow grades. Too low back pressure will result in insufficient plasticizing and melting of ALFATER$^{XL®}$, which in turn will cause inhomogenous melt. Sufficient back pressure will provide effective plasticizing and melting of ALFATER$^{XL®}$, good melt homogeneity as well as good dispersion of additives and color masterbatches. However, high back pressure will also lead to longer plasticizing times.

Normally screws with small diameters can be run with higher screw speed than screws having large diameters. Screw speed should not be too high as this can cause excessive shear heating. It is recommended to apply sufficient level of back pressure in combination with medium screw speed (50 – 200 min^{-1}, depending on screw diameter).

HOLDING PRESSURE AND HOLDING TIME

Basically, flowability of ALFATER$^{XL®}$ is shear-dependent. Therefore, it is advisable that the mold is almost completely filled during the injection phase – so 98 % or more. Underfilled molds cannot be sufficiently and homogenously filled with holding pressure. The holding pressure applied, is predominantely for warpage minimization and 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 need normally longer holding times than thin-walled parts. Besides holding pressure, shrinkage can also be compensated by the use of the holding time. Longer holding times will lead to reduced 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.

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 degrees taper are desirable. Tapered or Z-pin sprue-pullers are recommended (Figure 4).

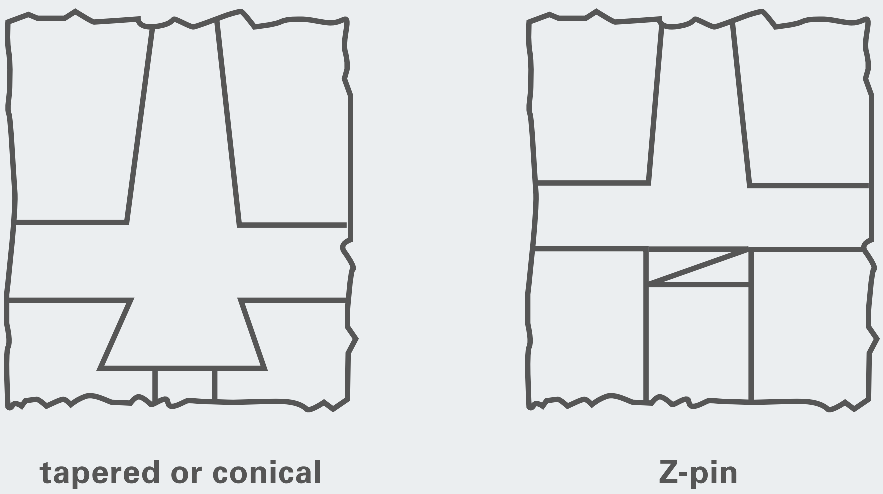


Figure 4: Recommended sprue-pullers for ALFATER$^{XL®}$.

Runner

Today, multi-cavity molds are often used in high volume applications. It is important that all cavities are filled in the same time and under the same pressure. Therefore, the runner should be balanced. If the runner system is not correctly balanced, cavities closer to the spure will be overpacked while cavities away further from the spure won't be completely filled (Figure 5).

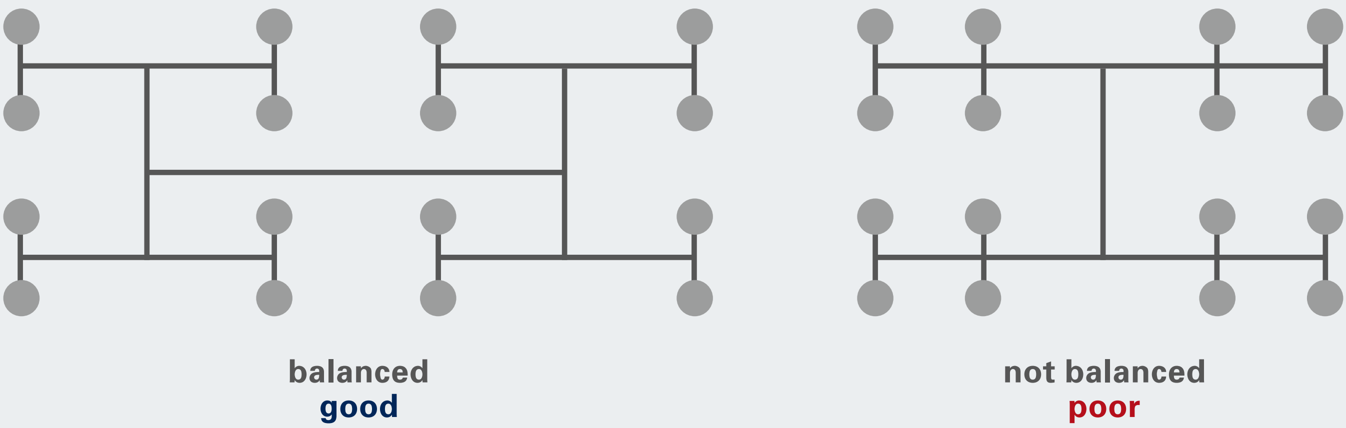


Figure 5: Example for balanced and unbalanced runner design.

The runners should be as short as possible and as smooth as possible without dead areas or sharp edges which could increase shear or material degradation. In addition the dimensions of the channels should be large enough to minimize the pressure drop in the melt.

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 6).

Figure 6: 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 when 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 a pin instead of 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 land 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 diamater 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 moulding simulations, e.g. Moldflow or Moldex3D for optimal mold design, gate location, and number of gates. Moldflow and Moldex3D data for ALFATER$^{XL®}$ grades are available upon request. Please ask your local local ALBIS Sales Office for details.

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 resisdence 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 unmelted 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 distrubtion through the complete runner system and to avoid burrs, sharp corners or dead areas as this can promote material degradation. To achieve sufficient shearing of the ALFATER$^{XL®}$ melt, the hot runner should have rather small channels.

Full round hot runner channels are preferred. The gate of the hot runner systems should be small enough for sufficient 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. For open nozzle systems the diameter can be significantly < 1 mm e.g. 0.6mm. However, open nozzle systems can cause drool, gate vestige and demolding problems due to the elastomeric properties of ALFATER$^{XL®}$. Therefore, open nozzles 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

Appropriate mold venting is necessary for efficient processing and good part quality due to the high injection speed required for ALFATER$^{XL®}$. 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.

Ejection

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 combined with 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. Furthermore, 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 to the mold surface.

USE OF REGRIND

Basically waste which is produced during injection molding can be recycled and used as regrind in new production. The maximum level of regrind which can be added to new production depends on several factors such as the part design and application. Therefore, it is advisable to define the specific feeding level of regrind for each individual injection molding process and part design. Regardless of this advice, a feeding level of 20 % regrind in new production is the general recommendation. The recycled material can be dry-blended (physically mixed) with prime (virgin) material. The regrind should be clean and free from any impurities. Furthermore, grinding of the scrap to a particle size comparable to the virgin granules is advisable for effective dry-blending. Drying of the recycled material is also recommended. Drying conditions are similar to prime quality, 2 – 4 hours at 70 – 80 °C in a dry-air dryer.

COLORING

Natural ALFATER$^{XL®}$ can be easily colored. The best choice would be a colored ALFATER$^{XL®}$ compound as this ensures the highest color consistency and batch-to-batch repeatability – please contact your local ALBIS Sales Office for more information on ALFATER colour compounds.

Alternatively polyolefin-based color masterbatches can be used. In this case, PP or PE color batches offer excellent compatibility with ALFATER$^{XL®}$ and are preferred. ALBIS offers a broad portfolio of PP and PE color batches e.g. from AMPACET. The viscosity of the color masterbatch should be in the range of a typical injection molding grade providing sufficient flowability. Typical feeding level of the batch is 1 – 5 %. However, the optimal dosage can vary depending on the product or application. It is therefore highly recommended to discuss optimal feeding level either with the color masterbatch producer or with the local TSAD (Technical Services & Application Development) of ALBIS PLASTIC GmbH.

Sufficient mixing of the color masterbatch with the ALFATER$^{XL®}$ melt 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 e.g. mixing head and/or static mixer further improves the dispersion of the color masterbatch.

Should you have any further questions regarding the processing or mold design for ALFATER applications, then please feel free to get in touch with your local TSAD engineer – contact details are available from the ALBIS website.

HEAD OFFICE

ALBIS PLASTIC GmbH
Mühlenhagen 35 · 20539 Hamburg
Telephone: +49 40 7 81 05-0
Fax: +49 40 7 81 05-361
info@albis.com · www.albis.com

ALBIS 1019