Contents

1 Products ................................................. 4
  1.1 Basic ACRYLITE® molding compounds
  1.2 ACRYLITE® Hi-Gloss for high-gloss surfaces
  1.3 ACRYLITE® Resist, impact-modified (zk)
  1.4 ACRYLITE® Satinice, light-diffusing (df)
  1.5 ACRYLITE® Heatresist and ACRYMID®,
      high heat deflection temperature under load
  1.6 ACRYLITE® LED, for backlit and edge-lit LED applications
  1.7 Colors
  1.8 Physical form and packaging

2 Physical Properties .................................... 8
  2.1 Thermal properties
  2.2 Mechanical properties
  2.3 Electrical properties
  2.4 Other properties

3 Technical Requirements .............................. 10
  3.1 Injection molded parts
  3.2 Injection molding process
  3.3 Injection molding machine
  3.4 Injection mold

4 Product Handling ...................................... 24
  4.1 Storage
  4.2 Material extraction
  4.3 Material transport
  4.4 Silo installations
  4.5 Drying requirements
  4.6 Miscibility and use of regrind

5 Processing on Injection Molding Machines ........ 28
  5.1 Plasticization
  5.2 Barrel melt temperatures
  5.3 Metering
  5.4 Back pressure
  5.5 Injection speed
  5.6 Hold/pack pressure
  5.7 Cooling time
  5.8 Mold temperature
  5.9 Injection compression molding

6 Injection Molding Defects, Causes and Remedies .... 32

Cover: © Eschenbach Optik GmbH
Introduction

ACRYLITE® molding compounds are thermoplastics based on polymethyl methacrylate (PMMA). The sum of their product-specific, chemical and physical properties make them well suited for manufacturing high-quality parts on injection molding machines.

Crystal-clear ACRYLITE® molding compound is free from haze and absolutely colorless. This makes it possible to produce optically clear injection-molded parts that transmit light with no absorption. The material’s unique colorlessness enables the production of particularly pure colors. ACRYLITE® is unsurpassed in terms of its resistance to weathering and aging. Even after decades of use, it shows no yellowing or surface defects. In addition, ACRYLITE® offers the highest surface hardness of all thermoplastics, and therefore the best scratch resistance.

Sustainability

ACRYLITE® can be completely recycled, either by chemical conversion back to its base materials or by direct reuse (regrind). These aspects, together with its environmentally sound production, contribute to the material’s sustainability. To prove this sustainability, we have subjected ACRYLITE® to a comprehensive life cycle assessment from cradle to grave. From production to recycling, the different environmental impacts of ACRYLITE®, including its greenhouse gas reduction effect due to the material’s longevity, have been determined and confirmed in this Life Cycle Assessment (LCA) in accordance with DIN ISO 14040ff. ACRYLITE® also makes an essential contribution to reducing the impact on the environment during the manufacturing process. According to recent calculations, the carbon dioxide equivalent is 3.8 kg CO₂ per kg of molding compound.
1. Products

The ACRYLITE® molding compounds described below are used for injection molding. You can find more products with other properties and fields of application in our Molding Compounds Product Portfolio brochure or on our website www.acrylite-polymers.com

1.1 Basic ACRYLITE® molding compounds

Basic ACRYLITE® molding compounds are available in several different grades. Depending on the grade, they offer different physical properties, such as ease of flow and heat deflection temperature. They can be processed by means of all conventional thermoplastic processing methods.

1.2 ACRYLITE® Hi-Gloss for high-gloss surfaces

Molding compounds of the ACRYLITE® Hi-Gloss product family are particularly suitable for injection molded technical parts. This material makes it possible to manufacture high-gloss, Class A surfaces. They are usually provided in opaque colors. The unique color Piano Black was developed especially for deep, jet black high gloss applications. The main applications are exterior automotive trim parts (e.g. pillar panels, spoilers, roof elements), and decorative trim in car interiors and exteriors, mirror housings, etc.

Depending on the grade, products of the ACRYLITE® Hi-Gloss family offer a range of higher heat deflection temperature under load, better flow and higher impact strength.

<table>
<thead>
<tr>
<th>Properties and behavior of basic molding compounds</th>
</tr>
</thead>
<tbody>
<tr>
<td>Key property</td>
</tr>
<tr>
<td>Mechanical strength</td>
</tr>
<tr>
<td>Long-term mechanical behavior</td>
</tr>
<tr>
<td>Hardness</td>
</tr>
<tr>
<td>Elongation at break</td>
</tr>
<tr>
<td>Heat deflection temperature</td>
</tr>
<tr>
<td>Flow</td>
</tr>
</tbody>
</table>
1.3 ACRYLITE® Resist, impact-modified (zk)

Impact-modified ACRYLITE® Resist molding compound is a thermoplastic based on polymethyl methacrylate (PMMA) that has been modified with an elastomer for greater impact strength. Like the basic ACRYLITE® molding compound grades, ACRYLITE® Resist combines very good weather resistance with excellent light transmission and clarity. At higher degrees of impact resistance, there is a reduction in rigidity and strength, while the elongation at yield and elongation at break are increased.

Compared with standard PMMA, impact-modified ACRYLITE® Resist molding compound offers increased stress crack resistance. Since there are different groups of impact-modified ACRYLITE® Resist molding compounds, only several of the grades are listed here in more detail as example. The behavior of the other groups is similar.

1.4 ACRYLITE® Satinice, light-diffusing (df)

Parts injection molded from ACRYLITE® Satinice have a light-diffusing effect that is based on embedded bead-like polymer particles. While delivering high light diffusion, they retain almost all of their light transmission. This makes such parts especially suitable for conventional light sources. Please consult the Molding Compounds Product Portfolio brochure for details of the available ACRYLITE® Satinice products and the relevant basic molding compound grades.

---

### Properties and behavior of ACRYLITE® Resist molding compounds

<table>
<thead>
<tr>
<th>Key property</th>
<th>Behavior of molding compound grades from zkP to zk6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mechanical strength</td>
<td>increasing</td>
</tr>
<tr>
<td>Impact strength</td>
<td>increasing</td>
</tr>
<tr>
<td>Hardness</td>
<td>decreasing</td>
</tr>
<tr>
<td>Elongation at break</td>
<td>increasing</td>
</tr>
<tr>
<td>Heat deflection temperature</td>
<td>decreasing</td>
</tr>
<tr>
<td>Flow</td>
<td>decreasing</td>
</tr>
</tbody>
</table>
1.5 ACRYLITE® Heatresist and ACRYMID®, high heat deflection temperature under load

The products of the ACRYLITE® Heatresist and ACRYMID® groups are crystal-clear molding compounds with different heat deflection temperatures under load.

ACRYLITE® Heatresist FT15 is a molding compound based on PMMA with a higher heat deflection temperature under load combined with improved flow. It is particularly suitable for injection molding applications with stringent requirements in terms of heat deflection temperature and flow. The special property profile offers benefits especially when it comes to designing parts with challenging wall thickness/flow path ratios (e.g. multi-component injection molding).

ACRYLITE® Heatresist hw55 is particularly suited for injection molding technical parts for applications subjected to high thermal and chemical stresses. It is a copolymer based on methyl methacrylate (MMA) with comonomer constituents. These provide a high heat deflection temperature under load for a PMMA molding compound, combined with particularly high chemical resistance and ease of processing.

1.6 ACRYLITE® LED, for backlit and edge-lit LED applications

This is a group of specialty molding compounds for efficient lighting engineering applications in combination with LEDs. There is a choice of products either for edge lighting with maximum transmission or for backlighting without any disturbing hot spots.

**For edge lighting**
Components made from the molding compounds of the LD range appear crystal-clear and transparent when unlit. These molding compounds have been optimized for edge lighting and for guiding light across differently sized areas. No additional diffusion films or microstructures are required on the component surface in order to achieve uniform light output over the entire surface.

**For backlighting**
Colored molding compounds produce uniform light distribution when backlit with strong LED light, combined with high light transmission, and without any disturbing hotspots (spots of light). These properties make it possible to reduce the spacing required between the cover and the LED light source. This allows easy optimization of the wall thickness of the component.
1.7 Colors

The absolute colorlessness of crystal-clear ACRYLITE® molding compounds makes it possible to obtain extremely pure and vivid colors. Transparent hues show no haze and provide optimal light transmission combined with intense color.

Whereas translucent colors are highly light transmitting with good diffusion effects, the attraction of opaque colors lies in their saturation and high-value appearance. The light transmission of the individual color varies with the wall thickness.

Only the highest quality colorants are used to ensure that the colors do not fade or bleed and are protected against the adverse effects of processing temperatures.

1.8 Physical form and packaging

ACRYLITE® molding compound is supplied as pellets of uniform size. Our standard form of packaging is the 1500 lb cardboard box with polyethylene lining. Further types of packaging and silo delivery are available upon request.

When properly stored, all types of packaging ensure that moisture absorption is kept to a minimum.

You can obtain more information about the available colors on request. Send an email to: www.cyro.polymer@evonik.com
2. Physical Properties

The many different grades of ACRYLITE® molding compounds are distinguished by their physical properties. Besides the specific characteristics of PMMA described here, you will find detailed material data on all product grades in our Molding Compounds Product Portfolio brochure or on our website www.acrylite-polymers.com. Our molding compounds are also included in the Campus material database, together with comprehensive values and graphs (www.campusplastics.com).

2.1 Thermal properties

Like all thermoplastics, basic ACRYLITE® molding compounds and the parts made from them change their physical state in the event of pronounced temperature changes. The solid state extends from the beginning of the softening range, which is between 158 °F to 212 °F (70 °C to 100 °C) for the basic molding compounds, depending on the grade. This is followed by the thermoelastic state, which extends to about 338 °F (170 °C). From about 338 °F (170 °C), ACRYLITE® molding compound becomes thermoplastic, but melt temperatures greater than 392 °F (200 °C) are usually required for injection molding. The melt volume rate MVR (230 °C/3.8 kg) is a rheological value that describes melt flow behavior at 446 °F (230 °C).

Measurable thermal degradation of ACRYLITE® molding compounds first takes place at temperatures of almost 572 °F (300 °C). That means they can be processed within a wide temperature range without yellowing or decomposition.

2.2 Mechanical properties

The mechanical behavior of basic ACRYLITE® molding compounds do not change substantially from a very low temperature up to 176 °F (80 °C). Parts made from ACRYLITE® show the highest surface hardness of all thermoplastics. As a result, they offer good scratch resistance, which means they remain visually appealing even after prolonged handling.

2.3 Electrical properties

Electrical engineering opens up a wide field of application for ACRYLITE® molding compound. The tracking resistance is so very remarkable that ACRYLITE® molding compound are rated in the highest class according to DIN VDE 0303 with a CTI value of >600.

The high surface resistivity is retained even after exposure to an electric arc. Owing to their good dielectric strength, parts made from ACRYLITE® molding compound are also used in high voltage systems. Friction may cause static charging of parts made from ACRYLITE® molding compounds, which may make them more likely to attract dust. This can be avoided in the long term by regular cleaning, e.g. with an antistatic plastics cleaning agent.
2.4 Other properties

Parts made from ACRYLITE® molding compound absorb very little water. Even after prolonged storage in water, water absorption is normally only about 2%, and even less under normal conditions. This does not affect the electrical values, dimensional stability, strength and other physical properties to any noticeable extent.

Water vapor permeability is within the normal range for thermoplastics. The permeation coefficient is $4.5 \times 10^{-10} \text{ g cm/cm² h Pa}$.

The Flash Ignition Temperatures range from 720 °F to 750 °F (382°C to 399°C) and Self Ignition Temperatures all measure 850°F (454°C) according to ASTM D1929. The ignition temperature of all ACRYLITE® molding compounds is 806 °F (430°C) according to DIN 51794. They burn without soot and do not release any toxic gases. Fire behavior is classified as B2, normally flammable, in accordance with DIN 4102.

Underwriter Laboratories classifies ACRYLITE® molding compounds in class UL 94-HB and are (f1) outdoor rated.
3. Technical Requirements

Many thousands of articles, from parts weighing a few ounces (grams) to objects weighing several pounds (kilograms), are produced on injection molding machines for a wide variety of applications. ACRYLITE® molding compounds are user friendly and easy to process on conventional machines.

The requirements for achieving the best and most economical product results are as follows:
1) Design the plastic part to optimize the material and manufacturing process and select the most suitable ACRYLITE® molding compound grade
2) Skillful handling and drying of the molding compound
3) A sufficiently powerful injection molding machine expertly designed, and precision built injection mold
4) Correct processing techniques, skillful post treatment, clean production rooms, and equipment

3.1 Molded part

The 2006 VDI guideline "Gestalten von Spritzgießteilen aus thermoplastischen Kunststoffen" (Designing injection-molded parts made from thermoplastics) gives a good summary of all important design points. In principle, the statements it contains also applies to parts made from ACRYLITE® molding compounds.

3.1.1 Dimensioning and design

First of all, it needs to be clarified if the planned part will be exposed to short term or permanent mechanical stress. The values presented in the table of properties, divided by a factor of 2 to be on the safe side, generally apply to short term mechanical stress. For exposure to permanent mechanical stress, the permissible stress should not exceed 1450 psi (10 N/mm²) wherever possible. These values apply under normal conditions.

If a part is exposed to higher temperatures, the safety factor should also be increased because the mechanical values deteriorate at elevated temperatures.

If components are subject to the influence of solvents or plasticizers, their chemical resistance should be examined beforehand and the safety factors correspondingly elevated.

Thin-walled parts
Wall thicknesses of less than .04 in (1 mm) are very demanding in terms of processing and design of the molded part. Thin-walled parts often show deflection at temperatures below the stated values because of their higher molecular orientation due to rapid cooling. This may lead to warpage at service temperatures somewhat below the stated heat deflection temperature.

However, nowadays it is still possible to achieve good results with wall thicknesses of less than .04 in (1 mm), using optimized processing techniques (e.g. injection compression molding, special temperature-controlled molds or rapid injection).

Thick-walled parts
Wall thicknesses of 1 in (30 mm) and more can be injection molded with impeccable results, given sufficient time and technical effort. Injection compression molding combined with a mold temperature that can be varied throughout the injection molding cycle offer advantages for producing extremely thick-walled parts.

Notch Effect
Since PMMA is sensitive to notch effects due to its specific properties, sharp edged transitions should be avoided when designing the shape. Even small radii at transition sites considerably increase resistance to breakage. Since gate marks, especially pin gate marks, may also count as notches, they should if possible be located at locations on the part that are exposed to the least amount of stress.

Weld Lines
Weld lines occur when two melt flow fronts meet when the injection is carried out around cores (all kinds of apertures), or when there are local differences in flow resistance. Apart from causing optical defects, they may also impair the mechanical stability of the molded part.

If the position of the injection point is well chosen, weld lines can be moved to less visible areas of the molded part.

Processing conditions also play a role. Weld lines are usually reduced by high mold and melt temperatures, high injection pressure, and sufficiently high injection speed.

Using special processing methods, weld lines behind apertures and cut outs can be prevented or minimized.

(Details of chemical resistance can be provided on request).
One way to do this is by locally adjusting the injection mold temperature. The mold is then only briefly heated to a high temperature in the weld line area prior to injection. In exceptional cases, apertures must be machined after injection molding.

**Metal Inserts**

Metal inserts encapsulated by injection molded in parts made from ACRYLITE® molding compounds may be possible depending on the application. The concern to watch out for here is increased stress around the insert due to mold shrinkage and different coefficients of linear thermal expansion. Preheating the inserts has a favorable influence on the overall stress level of the molded part.

Inserts should not have sharp edges, since this may increase notched stress in the plastic. If stress cracking occurs, it is advisable to use an ACRYLITE® Resist molding compound.

Normally, it is better to insert metal parts afterwards, e.g. by bonding or ultrasound technology.

**Draft angles**

Demolding should be supported by draft angles on the molded part. A draft angle of at least one degree should be provided. A well polished mold surface or defined surface texture will also help improve demolding.

**Undercuts**

Undercuts are not recommended in parts made from ACRYLITE® molding compounds since they may lead to breakage of the part during demolding. Molds with sliders are a better way to obtain undercuts.

3.1.2 **Mold filling**

The flow curves shown above help to estimate the mold filling for materials under consideration using the wall thickness and the flow path to determine the most suitable material for the application. The table above shows you examples of flow curves for basic ACRYLITE® molding compounds.

On request, we will be pleased to provide flow curves for other material grades.

For reliable mold filling, it is advisable to select wall thicknesses that are slightly larger than those based on the flow curves. The length and diameter of the runners must also be taken into account.

Our product information sheets contain statements on the melt and cylinder temperatures for the individual molding compound grades.

If complex injection molds are used, it is cost effective to simulate the filling process using suitable simulation software (e.g. Moldflow®). EVONIK can provide the necessary material data sets and offer technical support for this.
3.2 Injection molding process

Various injection molding systems, variants of these and special techniques can be used to process various ACRYLITE® moldings compounds. The following processes are conventionally used in the industry:

- standard injection molding
- injection compression molding
- multicomponent injection molding
- film backmolding/in-mold coating
- injection blow molding

The suitable process is determined according to the requirements to be met by the component (geometry, function, and functional implementation in the component, quality, etc.).

3.3 Injection molding machines

ACRYLITE® molding compounds can be processed on all conventional injection molding machines. A machine with state-of-the-art controls and process data recording should be used to manufacture parts for demanding applications. The choice of drive system depends on the parts to be produced and the quality requirements.

In general, an electric drive system can offer advantages such as precise pressure and stroke control, and energy consumption. However, electric drives do not cope well with thick-walled parts and the related long hold pressure or compression times. Hydraulic core pull is often difficult to achieve with electric machines. Hydraulic drives offer a proven system with very high flexibility in terms of core pull, as well as long hold pressure and compression times. If correctly maintained, hydraulic machines also offer a high standard of cleanliness. Hybrid machines (hydraulic clamping unit, electric plasticizing unit) combine the benefits of both drive systems.

3.3.1 Clamping force

For the standard injection molding process using ACRYLITE® molding compounds, a clamping force of approx. 350 kg/cm² or 2.5 tons/in² is sufficient in relation to the projected surface area of the molded part for a ratio of flow length/wall thickness <100/1. A clamping force of approx. 700 kg/cm² or 5 tons/in² is sufficient in relation to the projected surface area of the molded part for a ratio of flow length/wall thickness >100/1. The required clamping force can be reduced by about one third if a complete injection compression molding process is used.

If the clamping force is too low, this may lead to excess flash on the molded parts. Prolonged injection molding at inadequate clamping force may lead to mold damage. Excessive clamping force is usually not harmful. Particularly in special processes (injection molding of thin-walled parts with high filling speeds), higher clamping forces will be required.

3.3.2 Plasticizing unit

ACRYLITE® molding compounds can usually be processed using standard plasticizing units for thermoplastics.

Non-return valve

Screws for processing ACRYLITE® should be equipped with a ring-shaped non-return valve. Three-wing valves have proved suitable. The design of the non-return valve and the screw tip is meant to ensure good material throughput, to prevent material accumulations and the resulting black particles. Ball check valves are...
not considered suitable for processing ACRYLITE® molding compounds due to higher material shear.

**Screw geometry and plasticizing cylinder**
Our proposed zone divisions are consistent with standard screw recommendations. For processing PMMA, a reduction of the compression zone to 3 × D can additionally decrease the danger of air intake. In general, we recommend consulting the machine manufacturer regarding screws that are specially developed for PMMA. ACRYLITE® and ACRYLITE® Resist polymers can be processed on commercially available injection molding machines with general purpose screws, provided that the compression ratio is below 2.5:1. To maintain reasonable residence times and minimize shear degradation, the shot size should range from 40 to 60% of the barrel capacity. Standard screws and cylinders suitable for other thermoplastics (e.g. PC, ABS, etc.) can also be used.

**Special vented plasticizing unit**
In general, 5-zone vented screws with an open vented cylinder can be used to manufacture parts from ACRYLITE® molding compounds. The following points should be taken into consideration:

- the maximum metering stroke that can be used is limited
- risk of melt contamination by particles or degradation in the vent zone
- the temperature profile must be adjusted; a ~50°F (10°C) higher temperature should be set in the vent zone.

If a vented plasticizing unit is used, there is no need to dry the molding compound before the injection molding process. This can also compensate for a higher moisture level in the compound, e.g. due to incorrect storage. A vented screw is recommended in particular when processing products with a higher water content or proportion of inorganic, platelet-shaped fillers (e.g. certain pigments). Otherwise, the polymer chain or the fillers may require prolonged drying in a desiccant-type dryer.

---

### Injection Molding Screw Design Guidelines

<table>
<thead>
<tr>
<th>Item</th>
<th>Guideline</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Screw geometry without venting</strong></td>
<td>Screw length = 20D</td>
</tr>
<tr>
<td>Feed zone</td>
<td>L = 12 × D (60%)</td>
</tr>
<tr>
<td>Flight depth in the feed zone</td>
<td>h = 0.15 × D</td>
</tr>
<tr>
<td><strong>Compression zone</strong></td>
<td>L = 4 × D (20%) (better 3D) (core progressive)</td>
</tr>
<tr>
<td><strong>Metering zone</strong></td>
<td>L = 4 × D (20%)</td>
</tr>
<tr>
<td>Flight depth in the metering zone</td>
<td>h = 0.07 x D</td>
</tr>
<tr>
<td>Compression</td>
<td>2.1</td>
</tr>
<tr>
<td>Pitch</td>
<td>1D</td>
</tr>
</tbody>
</table>

### Screw Steel Recommendations

<table>
<thead>
<tr>
<th>Common steel types for cylinders</th>
<th>Material-no.</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.8519</td>
<td>31CrMoV9</td>
<td></td>
</tr>
<tr>
<td>1.8550</td>
<td>34CrAlN7</td>
<td></td>
</tr>
<tr>
<td>screws</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.4122</td>
<td>X35CrMo17</td>
<td></td>
</tr>
<tr>
<td>1.4057</td>
<td>X22CrNi17</td>
<td></td>
</tr>
</tbody>
</table>

3.4 Injection mold

The mold design and accurate mold production are crucial for the quality of a molded part.

3.4.1 Runners

The best cross-sections for runners are circular or trapezoidal. Elliptical or rectangular cross-sections are less favorable. (Fig. 2)

This is clearly shown by the example of a molded part with a gate at the end and a trapezoidal runner. (Fig. 3)

3.4.2 Cold runner gate

When processing ACRYLITE®, it should be taken into consideration that these molding compounds have a higher viscosity than other amorphous or partially crystalline engineering plastics. That is why the thickest and shortest possible gate and an adequately sized runner and gate configuration should be chosen in the mold cavity.

The following gate systems should be considered, depending on the part geometries and mold systems to be used. To provide good results when filling the cavity and demolding the part, the gate cross-section should usually be about one third of the part’s wall thickness, and half the wall thickness for thick parts.

Sprue gate (Fig. 4)

This connects the injection nozzle directly to the molded part over an increasingly large cross-section. The sprue gate should have a draft angle of at least 2 to 3 degrees, although a larger angle provides ease when demolding of the sprue.

Sprue gates are used for simple, thick-walled and precision molded parts.

If the sprue gate is not located directly next to the molded part, a cold slug well should be installed opposite the gate to collect the colder melt that first emerges from the nozzle. An undercut ejector can be installed here that can be used to extract the sprue from the bushing. (Fig. 5)

Special forms for extremely thick-walled parts (lenses, prisms...)

It is important for the entire gating system to be adequately sized in order to obtain high quality lenses.

During the hold pressure phase, which may take several minutes, depending on the wall thickness of the molded parts, enough melt has to be fed through to reduce volume shrinkage, and prevent the resulting sink marks and voids. The thickness of the thinnest cross-section should be one third to a half of the largest wall thickness.

The gate thickness is more important than its width, i.e. a narrow, thick gate cannot be replaced by a wide, thin gate.

Sharp edges at the runner and on the molded part should be avoided. The radii should be sufficient to prevent jetting and melt flashover.

To keep material losses to a minimum, it is advisable to place the mold cavities as close as possible to the (primary) runner. (Fig. 6)
Fig. 4: Sprue gate  
1 = Mold cavity  2 = Gate  3 = Sprue bush  4 = Sprue

Fig. 5: Sprue with collecting mouth for the cold slug  
1 = Injection part  2 = Ejector with extraction claw

Fig. 6: Runner system for a 0.551 inch thick lense

Section A-A

Detail B
**Pin gate**

This type of gate is often chosen for multi-cavity injection molds because the sprue is automatically removed during demolding, with no need for post treatment.

The gate vestige produced on the surface of the molded part can often be concealed from sight with proper gate location.

*Fig. 7* shows an injection mold with 12 cavities for thin-walled lenses with a pin gate.

It should also be taken into consideration that the molded part shows reduced mechanical strength near the point where the sprue is removed.

*Fig. 8* shows a sprue puller tool for sleeves that are injected at the end through four pin gates each.

These gates should not be undersized because this will require excessive injection pressures. This causes the material to encounter excessive shear rates and the molded part cannot be sufficiently filled. For small parts, a diameter of 0.024–0.028 in (0.6–0.7 mm) is sufficient. PMMA has relatively high cohesive forces, this may cause jetting in the mold cavity causing undesired effects.

In *Fig. 9*, the pin gate is installed on the side, directly on the molded part.
Fan Gate

This type of gate is preferred for molded parts such as rulers, tail light lenses or trailer windows that must not have any marks on the visible surface.

To avoid one side loading of the injection mold and clamping unit, two mold cavities should be provided for reasons of symmetry. (Fig. 10)

This also has the advantage that the runner can be appropriately sized to meet the requirements. The thickness of the gate is more important than its width. (Fig. 11)

To ensure uniform filling of the mold cavity, all sharp edges should be given a reasonable radius to prevent jetting, air inclusions due to melt flashover, or weld lines.

For longer parts such as rulers, sheets or automotive pillar panels, it is ideal to place the gate at the end of the part.

This causes longitudinal molecular orientation, which increases the strength of the part and promotes its resistance to flexural stress. (Fig. 12)
Tab gate for large parts
This special variant is used for parts with a large surface area, such as panels or covers, to ensure uniform melt distribution. (Fig. 13)

Tunnel gate
With tunnel gates, the molded part is automatically separated from the runner during demolding. The runners are installed close to the molded parts at the parting line, and are connected with the molded part by a tunnel that leads at an angle from the parting line to the mold cavity. (Fig. 14)

When the mold opens, the molded part initially remains connected to the runner on the moving mold half, and is severed when the ejector moves forward. The sprue remains connected to the runner and has to be pulled out of the tunnel together with the runner. In other types of construction, the sprue is severed when the mold starts to open.

With this type of gate, molded parts can only be injected from the side.

Due to the natural brittleness of PMMA, this often produces small fragments that settle on the mold surface when the next shot is injected, and may lead to defects.

The high shear rates during injection around the gate may also lead to processing issues. Tunnel gates are therefore only suitable for thin walled parts injection molded from ACRYLITE® molding compounds.

3.4.3 Hot runner systems
ACRYLITE® molding compounds can be processed with hot runner systems either with open or with shut-off nozzles. These systems usually ensure low pressure losses with clearly defined, streamlined runner cross-sections. (Fig. 15)

Externally heated nozzles should be used for processing ACRYLITE® with hot runner systems. Internally heated (torpedo) hot runners pose problems because of their structure (annular cross-section) and the related pressure losses and color shifts. Pressure losses call for a higher temperature and may lead to shearing, polymer degradation and gate marks.

Designing hot runner systems always entails balancing shear stress on the melt during injection and the purging effect of the hot runner system. To avoid an unnecessarily long dwell time for the melt, the melt volume in the hot runner should be no greater than the shot weight.

In order to prevent local temperature peaks and the related damage to the material, symmetrically arranged heat conductors should be installed to ensure uniform and precise temperature distribution in the hot runner system.

In general, it should be ensured that the melt is conveyed gently through the runners. These should have large cross-sections, be free of corners and edges, and convey the melt by means of deflecting elements designed to provide optimized flow, as far as possible without stagnation zones.

Care should be taken to ensure a natural melt balance in multi-cavity molds. If the flow paths to the cavity are short, the melt volume in the hot runner can be kept to a minimum.

It is recommended to calculate and optimize the entire system (hot runners, hot runner nozzles and gate) with the hot runner manufacturer, and to perform a mold filling simulation (e.g. Moldflow®) based on the material data.

Needle valve system
When dimensioning the nozzle opening or shut-off nozzle, care should be taken not to select too small a nozzle opening, since this may damage the material at high shear speeds. Cylindrical shut-off systems are recommended. However, conical shut-off systems (annular split) can also provide good results.
Precise thermal separation of the injection mold, hot runner (especially the hot runner nozzle) and the cavity must be ensured. If too much thermal energy is transferred to the cavity, this may have a negative effect on this area of the molding (sink marks, optical defects). Under practical conditions, separate annular temperature control, including cooling lines, around the hot runner nozzle has proved to allow a wider processing window.

When processing ACRYLITE® molding compounds, it is possible to locate the sprue on the molded part in a visible area by using hot runner needle shut-off mechanisms. The requirements in terms of sprue visibility should be borne in mind. The sprues should be as unobtrusive as possible. Instead, the needle front area should be polished, slightly convex and protrude a few tenths of a millimeter (approx. 0.004 in [approx. 0.1 mm]) into the part.
3.4.4 Shrinkage of ACRYLITE® and ACRYMID® molding compounds

A fundamental distinction is made between plastics in line with standards for molding shrinkage, post-shrinkage and overall shrinkage. These shrinkage values are determined in accordance with ASTM D 955. Since ACRYLITE® and ACRYMID® molding compounds are amorphous materials, the molding shrinkage is relatively low and post-shrinkage is negligible.

Both types of shrinkage are therefore taken together. Generally, the molding shrinkage is different in the direction of flow and at right angles to the direction of flow. Somewhat higher shrinkage values are therefore measured in longitudinal direction for ACRYLITE® molding compounds.

The processing parameters also have a major influence on shrinkage. The holding pressure is crucial. The higher the effective holding pressure, the lower the resulting shrinkage.

The table above shows shrinkage tolerances for various products, listed according to processing conditions.

3.4.5 Venting the mold

The air present in the cavity must be able to escape during the injection process. This can usually be done at the parting line or at the ejector. If this is not possible, the air is compressed at the end of the flow path and causes black marks on the molded part ("diesel effect") and on the mold surface. Permanent injection molding under these conditions may damage the surface of the cavity. To ensure reliable mold venting, venting channels in the parting surface have proved useful. A channel that is .0079 to .0394 in (0.2 to 1 mm) deep can be milled in the parting surface of the mold. In the transition to the cavity, an opening that is about 0.0006 to 0.002 in (0.015 to 0.051 mm) deep and 0.2 to 0.4 in (5 mm to 10 mm) wide should be inserted. It is advisable to start with a venting channel of less depth in order to prevent any flashing.
3.4.6 Demolding the molded part

PMMA is an amorphous polymer with very low shrinkage. Owing to its mechanical properties, care must be taken not to deform the part during removal from the mold. Otherwise the part may break.

Apart from the previously mentioned draft angles (at least one degree) on the molded part and the gate system, it should be ensured the ejector applies loads homogeneously. Contoured ejectors have proved especially suitable. The draft angles in demolding direction should also be adequately polished.

In particular with molded parts that may generate a vacuum during demolding because of their shape, it has proved expedient to use some air for easier separation. This can be introduced by porous sintered metal inserts, for instance. It is not advisable to demold undercuts on parts made from ACRYLITE® molding compound, since this may cause the part to break. Molds with sliders should be used to produce undercuts.

The maximum demolding temperatures for the ACRYLITE® part should not be substantially exceeded.

(See diagram above)
3.4.7 Tool steel and surface treatment

Basic ACRYLITE® molding compounds do not release any corrosive substances during injection molding that might damage the mold surface. Conventional types of tool steel can therefore be used for the cavity inserts. What is important is the suitability for polishing. For this reason, very pure grades of chrome-containing steel with a homogenous morphology are used. These are manufactured either by a vacuum remelting or an electroslag remelting. (See table above)

Conventional types of steel (e.g. 1.1730) can be used for the master form of the mold. In some special cases, such as when using certain ACRYLITE® specialty molding compounds, the molding may tend to adhere more strongly to the mold surface. In this case, it has proved effective to coat the mold surface with TiN or CrN.

3.4.8 Mold temperature control

The influence of mold temperature on the properties of the finished part is often underestimated when processing ACRYLITE® molding compounds. If the melt is injected into a mold that is too cold, this makes it more difficult to fill the mold, and may lead to high cooling stress, warping, strong orientation and sink marks. Heating/cooling channels should be sized and positioned to keep all parts of the molding at a constant temperature. It should also be borne in mind that more heat needs to be dissipated for thick parts of moldings than for thin ones. The closer the channels are to the surface of the molded part, the more effective the temperature control is.

Shrinkage of parts with large wall thickness differences can be compensated by adjusting different parts of the mold to different temperatures. Usually, the side of the molding that tends to deflect concavely is heated to a lower temperature, while the temperature on the other side remains the same or is higher.

Separately adjusted or pointwise application of a higher temperature where melts flow together (weld lines) may provide parts with better optical and mechanical properties.

ACRYLITE® molding compounds need mold temperatures of more than 140 °F (60 °C). In special cases, a temperature of up to 284 °F (140 °C) may be required. Molds can be brought to this temperature by means of pressurized water, oil or electric resistance heaters. In this case, both the mold and the entire temperature control periphery (heating and cooling equipment, media channels, couplings) must be designed for these temperatures. The heat that radiates from the outside of the mold may prevent uniform temperature control despite having installed the correct heating capacity. In this case, it makes sense to insulate the outside of the mold.

In special cases (e.g. high differences in the wall thickness of the molded part or highly accurate reproduction of micro and nanostructures), dynamic temperature control of the mold may be required. Various systems are available on the market for this purpose.

---

### Tool Steel Recommendations

<table>
<thead>
<tr>
<th>Material No.</th>
<th>Code name</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2083 ESU</td>
<td>X42Cr14</td>
</tr>
<tr>
<td></td>
<td>Stainless steel, very suitable for high-gloss polishing</td>
</tr>
<tr>
<td>1.2343 ESU</td>
<td>X38CrMoV5–1</td>
</tr>
<tr>
<td></td>
<td>Hard-wearing steel, very suitable for high-gloss polishing. Can also be used for large molds.</td>
</tr>
<tr>
<td>1.2767 ESU</td>
<td>X45NiCrMo4</td>
</tr>
<tr>
<td></td>
<td>Very tough steel, very suitable for high-gloss polishing</td>
</tr>
<tr>
<td>1.2764</td>
<td>X36CrMo17</td>
</tr>
<tr>
<td></td>
<td>Steel for very large molds, can be polished to a certain extent</td>
</tr>
<tr>
<td>1.2316</td>
<td>40CrMnNiMo8–6–4</td>
</tr>
<tr>
<td></td>
<td>Stainless, can be polished to a certain extent</td>
</tr>
<tr>
<td>1.2738</td>
<td>40CrMnNiMo8–6–4</td>
</tr>
<tr>
<td></td>
<td>Steel for very large molds, can be polished to a certain extent</td>
</tr>
</tbody>
</table>
A distinction is made between these depending on whether they are positioned (outside or inside the mold), and according to the energy transfer method:

• external process based on fluid heating and fluid cooling

• external process based on electric heating (resistance heating, inductive or IR heating) and fluid cooling

• internal processes based on electric heating (resistance or inductive heating) and fluid cooling

Common to all systems is the need to position a temperature sensor near to the cavity, which is required for controlling the heating and cooling cycle. When it comes to designing and choosing the appropriate system, we refer you to the toolmaking companies and providers of temperature control systems, who have the necessary expertise.
4. Product Handling

Special measures need to be taken as regards material handling and processing in order to manufacture parts and surfaces that satisfy high optical requirements.

4.1 Storage

Like many other thermoplastics, ACRYLITE® molding compounds are hygroscopic. This means that the pellets absorb ambient moisture, depending on the storage conditions. Depending on their moisture content and the processing conditions, this may lead to defective moldings (bubbles, striation, matte surface, etc.).

Evonik’s packaging containers (cardboard boxes with PE lining, etc.) are designed to help prevent the pellets from absorbing moisture. However, to ensure optimal processing, we advise predrying the pellets.

The following storage conditions ensure maximum storage period with negligible moisture absorption:
- warehouse with constant climatic conditions
- open containers only shortly before use
- reseal containers (airtight) after opening.

4.2 Material extraction

In transparent ACRYLITE® components with high-gloss surfaces, even the smallest contamination during material handling and transport leads to visible defects in the injection molded part.

A key requirement for producing highly transparent moldings of impeccable quality using ACRYLITE® molding compound is therefore absolute cleanliness when preparing and processing the pellets.

A few comments concerning the handling of containers:
- Due to the electrostatic charge of the PE box liner, visible contamination should be removed from containers outside the processing area before they are opened.
- The PE liner should be cut open with a knife to prevent torn-off pieces of film from contaminating the pellets.
- Stainless steel scoops are recommended for removing small amounts of pellets. Aluminium scoops are not suitable for ACRYLITE® pellets because of possible metal fines.
- Opened containers should be resealed immediately after removing the pellets.

4.3 Material transport

Cleanliness is very important when handling the pellets.

Indications for manual transport of pellets:
- Only remove pellets in a clean and dust-free environment
- Use only stainless steel containers
- Use the transport containers for ACRYLITE® only. Never switch from other thermoplastics to ACRYLITE® without thoroughly cleaning the containers (if possible by washing).
- Avoid unnecessary refilling into different containers
- Label pellets that have been filled back from machine hoppers, due to the risks of cross contamination
- Do not reuse spilled pellets.

Recommendations for pneumatic conveying of pellets:

We recommend removing material from containers using a suction tube as shown below (Fig. 16).
Indications for designing pneumatic conveying units:

• The conveying air (air coming into the suction tube) must be filtered.
• The PE bag should be fastened to the suction tube so as to be airtight.
• Never remove material by suction from open containers!
• Use filters with a pore width of ≤ 5 µm.
• All tube parts and connectors that enter into contact with the pellets must be made of stainless steel.
• Aluminum tubes and hoppers, connectors made of copper etc. are unsuitable.
• Even the smallest contamination (e.g. fines from feed pipes) impair brilliance and optical properties.
• High conveying speed or low radius pipe bends increase PMMA abrasion (fines), which may later be visible in the molded part.
• If different thermoplastics are to be processed, it makes sense to use a separate conveying unit for ACRYLITE®. This saves cleaning costs and time, and prevents contamination with other material.
• If a conveying unit is to be switched from another plastic to ACRYLITE®, it must be thoroughly cleaned (dismantled, washed and dried).

Fig. 16: Extracting material from individual bags using a suction conveyor. To avoid damage to the PE inlet, it is advisable to attach a protective tip to the sharp end of the suction tube.
4.4 Silo installations

It is very important to choose the best material for storing ACRYLITE® pellets when installing silos. Since PMMA is very hard and therefore has an abrasive effect, plastic materials should not be used for silos. The same recommendations apply as for pneumatic conveying units.

Additional instructions for silo installations:
• The silo filling line and all feed pipes from the silo to consumers (e.g. drier, machine etc.) should be made of stainless steel.
• The recommended radius for all pipe bends in the conveyor lines is 10 D.
• Ensure that no seals protrude into the pipes or can come into contact with the pellets.
• Conveying air should be free from oil, dry and cleaned using a ≤ 5μm filter.
• It is advisable to feed dry air into the silo exit to prevent condensation in unfavorable weather conditions.
• To obtain uniform product output, the cone at the silo exit should not be too flat. An angle of 60° has proved suitable. At lower angles, there is a risk of core flow. Product fines may then accumulate at the sides, enter the product stream at intervals and cause conveying and processing problems. Such problems may occur especially when emptying the silo.

4.5 Predrying

ACRYLITE® molding compounds can be processed with perfect results if they are completely dry. High moisture levels can lead to defects in the molded part (striation and bubbles). This is due to water vapor that forms at processing temperature. ACRYLITE® molding compounds are packaged with a low degree of residual moisture, and can therefore be processed without predrying immediately upon supply and after brief storage. However, if they are exposed to unfavorable weather conditions during transport and storage, humid air may have diffused into the packaging, making it necessary to predry the product.

Cleanliness during drying
Experience while working with many customers has shown that drying entails a high risk of contamination.

Even microscopically small dirt particles lead to inclusions in molded parts. Metal fines from aluminum pipes, for example, cause glittering specks, and traces of other thermoplastics in PMMA lead to clouding.

Indications for optimal cleanliness during drying:
• The drier should stand in a clean room.
• The container should only be opened shortly before drying.
• Do not leave containers open after removing some of the contents.

4.5 Predrying

ACRYLITE® is incompatible with many other thermoplastics. Even the most minute traces of other plastics can lead to clouding or milky streaks in the molded part. Nor are various ACRYLITE® products always compatible with each other.

Drying conditions
The basic rule is to dry the compound at the highest possible temperature so that maximum moisture is removed as quickly as possible.

Individual molding compound grades require different drying temperatures due to the different heat deflection temperatures of the ACRYLITE® products.

Typical values for the maximum drying temperature: Vicat softening temperature minus 68°F (20°C.)
If the temperatures stated in the product data sheet are below recommended conditions, longer drying times will be needed. In unfavorable circumstances (environmental conditions, relative humidity), the residual moisture content of the compound is still too high even after several hours of drying.

The drying time depends on various factors:
- initial humidity of the pellets
- drying temperature
- required drying system
- required residual moisture content
- the type of ACRYLITE® compound being used

Basic molding compounds usually require a drying time of 2 to 4 hours. The recommended residual moisture content depends on the required quality and the processing technique used for the molded parts. *(See table above)*

4.5.1 Drying equipment
- **Drying cabinets**
  simple cabinets with or without circulating air are only to be used in exceptional cases.
- **Hopper drier**
  hopper driers installed directly on the machine have proved useful for molded parts with a small to medium shot weight. At high throughput, the drying time (throughput time) may not be sufficient.
- **Dry-air-drier**
  This type of drier provides the best drying results. Drying is performed by means of hot, dry air. Almost all of the driers available on the market are suitable for ACRYLITE® molding compound. The recommendations as regards type of construction and choice of material (stainless steel) in the section on material extraction and material transport should be borne in mind.

The dew point of the air in a desiccant-type drier is –4.0 °F to –22.0 °F (–20 to –30 °C). The special advantage is that the external climatic conditions are of no importance.

4.6 Miscibility and use of regrind
While different grades of ACRYLITE® molding compounds can be mixed in principle, this must be examined from case to case. Adding regrind of a separate grade (up to 20%) is only recommended for parts with modest quality requirements.

### Recommended residual moisture content

<table>
<thead>
<tr>
<th>Processing technique</th>
<th>Max. residual moisture content</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection molding with a vented cylinder</td>
<td>0.09–0.30%</td>
</tr>
<tr>
<td>Injection molding with a closed cylinder for less exacting applications</td>
<td>0.05–0.08%</td>
</tr>
<tr>
<td>Injection molding with a closed cylinder for exacting applications, such as the injection molding of thick-walled moldings or optical applications</td>
<td>≤ 0.04%</td>
</tr>
</tbody>
</table>

**Handling after drying**
The compound should be processed immediately after drying. Unclosed containers, open containers and hoppers pose problems in regards cleanliness and moisture absorption. The permissible moisture content may be exceeded after only one to two hours.

**Vented cylinders**
It is usually not necessary to predry the compound if injection molding machines with vented cylinders are used.
5. Processing on Injection Molding Machines

The following statements provide some basic guidelines for processing ACRYLITE® molding compounds on injection molding machines. The injection molding parameters must comply with the design of the molded part and the mold.

5.1 Plasticization

The plasticizing settings depend on the ACRYLITE® product being used. The relevant information can be found in our product information sheets.

5.2 Cylinder temperatures

The required melt temperatures for ACRYLITE® products range between 410 °F to 545 °F (210 °C to 285 °C) (depending on the molding compound grade including high heat grades).

The cylinder temperature setting is usually not identical with the actual melt temperature on injection molding machines. At high screw speeds, the real melt temperature is usually higher than the cylinder temperature setting. At lower screw speeds, it is usually lower. That is why it makes more sense to state temperature ranges rather than exact processing temperatures.

Due to the good heat stability of ACRYLITE® molding compound, brief interruptions in production and the higher thermal load these entail usually cause no harm. Only if the thermal load is unusually high may the melt become discolored, and decomposes with the typical odor formation in extreme cases.

The cylinder temperatures should be raised step by step from the hopper to the nozzle, with the nozzle temperature and at least the last section near to the nozzle being brought up to melt temperature. Due to temperature losses at the nozzle and the transfer of heat to the mold, a higher temperature may be needed in this area.

In the material feed zone (throat temperature), the temperature should be adjusted to between 122 °F to 185 °F (50 °C and 85 °C), and between 365 °F to 446 °F (185 °C and 230 °C) in the first cylinder heating unit. A higher temperature may be required in special cases (e.g. ACRYMID® molding compound). If the throat temperature is too low and the first cylinder zone too cool, this may cause noise during metering. If the temperature is too high, this may lead to bridge-building due to melted pellets and reduced flow in the feed zone.

For producing molded parts for exacting requirements, it has proved useful to connect an external temperature control device to the cylinder throat to ensure precise temperature control.
5.3 Metering

Metering should always be performed toward the end of the injection molding cycle and should be finished by the end of the cooling time. The screw speed must be selected as a function of the screw diameter. Smaller screws (up to a diameter of 1.57") can be operated at higher speeds (60–100 rpm), larger ones should be operated at lower speeds (20–50 rpm). Higher screw speed may increase the melt temperature due to the transfer of shear energy.

5.4 Back pressure

Back pressure prevents air intake during metering, and ensures homogeneous melt preparation. The effective back pressure in the molding compound should be between 1,160 and 2,900 psi actual (80 and 200 bar) at most. At higher back pressures, the melt temperature and mixing effect increases, whereas the plasticizing performance is reduced.

5.5 Injection speed

A high injection speed should be selected for thin-walled parts in order to fill the mold before the melt solidifies.

Thick-walled parts call for slow injection in order to fill the mold evenly and to prevent flow lines and creases in the molded part.
5.6 Holding pressure

The level and duration of holding pressure influences the properties of the molded part (sink marks, inherent stresses, etc.). To obtain the best possible quality of the molded part, the holding pressure should be as low as possible and be maintained for a prolonged period. In order to prevent inherent stress as far as possible, the holding pressure only needs to be as high and prolonged as is necessary to obtain satisfactory molding results. To achieve effective holding pressure, the gate system has to be adequately dimensioned so that it does not freeze before the molded part solidifies. Amorphous thermoplastics are usually processed at constant holding pressure. Reducing the holding pressure step by step may be advantageous for complex molded parts.

5.7 Cooling time

The cooling time mainly depends on the thickness of the molded part. The thickest wall of the part should be taken as an indication.

The required cooling time for a component in sheet form can be calculated using the formula below. To simplify calculation, heat transfer across the sides of the component is left out of the equation.

\[ t_k = \frac{s^2}{\alpha_{eff} \cdot \pi^2} \cdot \ln \left( \frac{8}{\pi^2} \cdot \frac{\theta_m - \theta_w}{\theta_e - \theta_w} \right) \]

where:
- \( s \) : thickness (mm)
- \( \alpha_{eff} \) : effective thermal diffusivity (m²/s)
- \( \theta_m \) : melt temperature during injection °C
- \( \theta_e \) : cavity surface temperature °C
- \( \theta_w \) : demolding temperature determined via the cross-section of the molded part °C
- \( t_k \) : cooling time (s)

The effective thermal diffusivity for PMMA is \( 0.103 \times 10^{-6} \) m²/s.

It is advisable to try to determine the minimum cooling time depending on the required quality of the molded part.

5.8 Mold temperature

The influence the mold temperature has on processing and on the properties of the finished part is often underestimated. For processing ACRYLITE® molding compound, this temperature should be 68 °F (20 °C) at most below the Vicat softening temperature. In most cases, though, a lower mold temperature is sufficient. A high mold temperature causes the melt to cool more slowly in the mold cavity and makes it possible to reduce the inherent stress level. More time must then be allowed for cooling. The mold surface reproduction on the ACRYLITE® molding is improved by a high mold temperature. This has proved particularly favorable for mold surfaces that are equipped with micro or nanostructures. The highest possible mold surface temperature should be chosen for very thick-walled parts in particular.

In most cases, both mold halves are operated at the same mold temperature. In the event of warpage of the molded part, it may be advantageous to adjust the two halves of the mold to different temperatures. Usually, the side that shows a tendency toward concave deflection is cooled, while the other mold half is heated or less intensively cooled.

The use of a mold temperature that can be variably controlled over the injection molding cycle has proved advantageous in several respects.
Partial cyclical heating of the mold may also reduce the visibility of weld line and flow lines.

If the cavity is cyclically controlled over the entire surface, this technology makes it possible to reduce cycle times when injection-molding thick-walled parts. The mold surface reproduction of macro, micro and nanostructures can also be improved, and the molding shrinkage can be reduced.

The best way to control the mold temperature as a function of cycle is to adjust the mold temperature during the cavity filling process to a value higher than the glass transition temperature of the polymer in most cases. The mold temperature then has to be reduced at least to demolding temperature.

5.9 Processing by means of injection compression molding

ACRYLITE® molding compounds are also suitable for the familiar injection compression molding processes. There are a large number of compression variants (e.g. expansion compression, compression of specific segments).

In general, the required clamping force can be reduced by one third for injection compression molding over the entire surface. Injection compression molding offers the advantage that the molded part is subjected to uniform pressure across the surface, either in addition to or instead of holding pressure, during the compression phase. This substantially reduces the level of inherent stress in the molded part. Resulting effects such as shrinkage of the molded part and warpage can be effectively minimized. Due to the optimal effect of compression pressure, the mold surface reproduction (e.g. of structures) can be additionally improved. Large differences in the wall thickness of the molded part can also be better controlled in an injection compression molding process. Here, care should be taken to prevent the compression pressure from becoming ineffective due to solidified, thin-walled parts of the molding.
# 6. Injection Molding Defects, Causes and Remedies

<table>
<thead>
<tr>
<th>Problems/Explanation/possible cause</th>
<th>Remedies</th>
</tr>
</thead>
</table>
| Mold is not completely filled       | • Check metering  
• Increase injection pressure  
• Increase injection time  
• Increase injection speed  
• Increase melt temperature  
• Increase mold temperature  
• Increase holding pressure  
• Increase back pressure  
• Change grade of material  
• Increase wall thicknesses |
| Not enough material                 | • Enlarge gate  
• Enlarge runner cross-section  
• Possibly enlarge runner |
| Pressure losses too high at the gate/sprue | • Larger cylinder or machine |
| Machine has inadequate shot weight  | • Increase mold temperature  
• Increase melt temperature  
• Increase injection speed  
• Change gate to improve flow ratios  
• Round-off transitions |
| Bubbles, voids                      | • Check metering  
• Reduce melt temperature  
• Increase holding pressure  
• Increase holding time  
• Reduce injection speed  
• Increase mold temperature  
• Increase back pressure  
• Ensure better predrying  
• Enlarge gate and possibly runner |
| Mold is not properly filled, shrinkage is not adequately compensated | • Increase mold temperature  
• Increase melt temperature  
• Increase injection speed  
• Change gate to improve flow ratios  
• Round-off transitions |
| Sink marks                          | • Check metering  
• Increase mold temperature  
• Reduce melt temperature  
• Increase holding pressure  
• Increase injection time  
• Increase injection speed  
• Reduce screw speed  
• Enlarge gate and possibly runner |
| Insufficient pressure or holding pressure to compensate for shrinkage | • Predry material  
• Reduce melt temperature  
• Increase back pressure  
• Change material grade |
| Surface shows fine grooves “Record” effect | • Increase melt temperature  
• Increase mold temperature  
• Increase injection speed  
• Reduce injection speed  
• Change and possibly reduce melt temperature (cylinder nozzle and hot runner)  
| Stick/slip effect of melt on mold surface | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure  
• Avoid sharp edges near the gate  
• Enlarge gate and possibly runner  
• Reduce dwell time in cylinder |
| Surface of molded part shows streaks and striping | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure |
| Matte spots (often around the sprue) | • Increase melt temperature  
• Check that nozzle unit is tight  
• If nozzle closure is mechanical, check that this works properly  
• Increase nozzle temperature  
• Increase mold temperature  
| Cold slugs                           | • Polish the nozzle; it must be conical so that the cold slug can be removed from the nozzle  
• Install pocket with ejector claw  
• Increase melt temperature |
| Differences in wall thickness, sharp-edged transitions | • Make wall thicknesses more uniform to achieve regular melt flow  
• Vent mold more efficiently |
| Creases and lines on the surface    | • Reduce injection speed  
• Increase mold temperature  
• Vary melt and hot runner temperatures  
• Round and polish gates  
• Enlarge gate and possibly runner |
| Melt jetting                         | • Install baffle plate |
| Flow lines, weld lines              | • Increase mold temperature  
• Increase melt temperature  
• Increase injection speed  
• Change gate to improve flow ratios  
• Round-off transitions |
| Recesses                            | • Vent mold more efficiently  
• Change gate to improve flow ratios  
• Round-off transitions |
| Matte spots (often around the sprue) | • Make wall thicknesses more uniform to achieve regular melt flow  
• Vent mold more efficiently  
• Change gate to improve flow ratios  
• Round-off transitions |
| Cold slugs                           | • Polish the nozzle; it must be conical so that the cold slug can be removed from the nozzle  
• Install pocket with ejector claw  
• Increase melt temperature |
| Chipping in gate area                | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure |
| Overloading                          | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure |
| Sprue breaks off                     | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure |
| Sprue is overloaded or not properly designed | • Polish sprue bushing  
• Make sure sprue bushing is conical and oversized in relation to the nozzle  
• Enlarge ejector claw or its undercut  
• Round Z-shaped ejector slightly to prevent sprue from breaking at this point  
• Prolong cooling time  
• Reduce injection pressure  
• Reduce holding pressure |
<table>
<thead>
<tr>
<th>Problems/Explanation/possible cause</th>
<th>Remedies</th>
</tr>
</thead>
</table>
| **Part breaks during demolding**  | - Reduce injection pressure  
- Reduce holding pressure  
- Increase mold temperature  
- Reduce mold opening and ejection speed |
| **Mold is overloaded or demolding temperature is too low** | - Remove undercut  
- Remove sharp mold edges  
- Improve draft angles  
- Improve mold polish  
- Additional ejectors  
- Enlarge wall thicknesses |
| **Mold is not optimally designed** | - Reduce back pressure  
- Predry molding compound  
- Reduce melt temperature  
- Reduce nozzle temperature  
- Use shut-off nozzle |
| **Drooling at the nozzle** | - Work with screw return stroke after metering  
- Allow nozzle to rest on mold |
| **Part remains attached on the nozzle side** | - Reduce metering  
- Reduce injection speed  
- Reduce holding pressure and time |
| **Rough surface** | - Polish surface in demolding direction |
| **Inadequate shrinkage of injected part** | - Increase cooling time  
- Reduce melt temperature  
- Improve mold cooling |
| **Rings around the sprue** | - Increase holding time  
- Reduce holding pressure  
- Increase nozzle temperature  
- Increase melt temperature  
- Increase mold temperature |
| **Melt is pushed onto material that has already cooled** | - Reduce nozzle temperature |
| **Threads on the sprue** | - Reduce injection speed  
- Increase mold temperature  
- Move gate to another place  
- Round sharp-edged transitions  
- Reduce depth of letters and/or round edges |
| **Air entrapment behind inserted recesses or raised parts** | - Increase feed temperature  
- Increase melt temperature  
- Control temperature of mold halves separately: cool the "hollow" side more than the other side |
| **Differences in wall thickness** | - Increase feed temperature  
- Increase melt temperature  
- Use a clamping jig when cooling the moldings |
| **Screw creaks (squeaks) loudly or stands still now and then** | - Increase feed temperature  
- Increase melt temperature  
- Vary screw speed |
| **Temperature in the feed zone is too low** | - Increase feed temperature  
- Increase melt temperature |
| **Screw motor is too weak** | - Chose stronger screw motor |
| **Brown and black particles** | - Purge (wash out) the cylinder and screw with our ACRIFIX® sp cylinder/barrel cleaning agent  
- Remove the screw in the event of stubborn soiling |
| **Contamination** | - Prevent contamination in the drier and/or conveyor system  
- Do not add regrind  
- Clean the vent section of the cylinder |
| **Black streaks, milky bubbles** | - Switch off machine return stroke  
- Increase back pressure  
- Vary the feed temperature  
- Check the temperature in the feed zone  
- Clean the cylinder, prevent contamination (other thermoplastics) |
| **Air intake, fluctuations in metering** | - Avoid long interruptions  
- Always remove/pull back screw when the machine is switched off |
| **Discoloration (yellow tinge)** | - Reduce melt temperature  
- Reduce screw speed  
- Reduce back pressure  
- Reduce dwell time  
- Reduce injection speed (frictional heating during injection)  
- Enlarge gate |
| **Material overheating and oxidative decomposition/air intake** | - Reduce melt temperature  
- Reduce screw speed  
- Reduce back pressure  
- Reduce dwell time  
- Reduce injection speed (frictional heating during injection)  
- Enlarge gate  
- Filter ambient air |
| **Contamination during predrying (soiling in drier, ambient air unclean)** | - Filter ambient air  
- Clean drier |
| **Included air bubbles at the edge of a molded part, burning at the end of the flow path** | - Reduce injection pressure  
- Reduce cylinder temperature  
- Change wall thicknesses of mold  
- Optimize predrying |
| **Unfavorable venting of mold** | - Prolong cooling time  
- Reduce mold temperature  
- Reduce melt temperature |
| **Parts are warped after demolding** | - Control temperature of mold halves separately: cool the "hollow" side more than the other side |
| **Dimensions of molded parts are too small** | - Reduce injection pressure  
- Reduce cylinder temperature  
- Install venting channels  
- Change wall thicknesses of mold  
- Optimize predrying |
| **Too much shrinkage** | - Reduce melt temperature  
- Reduce mold temperature  
- Increase holding pressure  
- Increase holding time  
- Inject more slowly |
<table>
<thead>
<tr>
<th>Problems/Explanation/possible cause</th>
<th>Remedies</th>
</tr>
</thead>
</table>
| **Dimensions of molded parts are too large** | • Increase melt temperature  
• Increase mold temperature  
• Reduce holding pressure  
• Reduce holding time  
Too little shrinkage | • Increase injection speed |
| **Flashing** | • Increase mold clamping force  
• Switch to holding pressure at an earlier stage (reduce metering)  
• Reduce holding pressure  
• Check mold parting line for damage |
| **Pressure too high inside the mold or clamping force adjusted to too low a level** | • Increase mold clamping force  
• Switch to holding pressure at an earlier stage (reduce metering)  
• Reduce holding pressure  
• Check mold parting line for damage |
| **Molding compound is too viscous** | • Use material grade with greater ease of flow |
| **Machine clamping force is too low** | • Use larger machine |
| **Striation, clouding or other inhomogenities inside the molded part** | • Remove other thermoplastics completely by washing  
• Purge (wash out) the cylinder and screw with our ACRIFIX® sp cylinder/barrel cleaning agent  
• Prevent contamination with other thermoplastics in the drier and/or conveying system  
• Do not add regrind |
| **Contamination or traces of other thermoplastics** | • Do not mix different grades of ACRIFIX® molding compound  
• Reduce injection speed  
• Increase mold temperature  
• Vary melt and nozzle temperatures  
• Round and polish gate  
• Reduce screw speed, increase back pressure  
• Use cylinder with better plasticizing properties |
| **Differences in wall thickness, sharp-edged transitions, poor plasticization** | • Completely remove other thermoplastics by washing  
• Purge (wash out) the cylinder and screw with our ACRIFIX® sp cylinder/barrel cleaning agent  
• Prevent contamination from other thermoplastics in the drier and/or conveying system  
• Do not add regrind |
| **Milky streaks** | • Completely remove other thermoplastics by washing  
• Purge (wash out) the cylinder and screw with our ACRIFIX® sp cylinder/barrel cleaning agent  
• Prevent contamination from other thermoplastics in the drier and/or conveying system  
• Do not add regrind |
| **Contamination with other thermoplastics** | • Completely remove other thermoplastics by washing  
• Purge (wash out) the cylinder and screw with our ACRIFIX® sp cylinder/barrel cleaning agent  
• Prevent contamination from other thermoplastics in the drier and/or conveying system  
• Do not add regrind |
Evonik Industries is a worldwide manufacturer of PMMA products sold under the ACRYLITE® trademark in the Americas and under the PLEXIGLAS® trademark in the European, Asian, African and Australian continents.

This information and all technical and other advice are based on Evonik’s present knowledge and experience. However, Evonik assumes no liability for such information or advice, including the extent to which such information or advice may relate to third party intellectual property rights. Evonik reserves the right to make any changes to information or advice at any time, without prior or subsequent notice.

EVONIK DISCLAIMS ALL REPRESENTATIONS AND WARRANTIES, WHETHER EXPRESS OR IMPLIED, AND SHALL HAVE NO LIABILITY FOR, MERCHANTABILITY OF THE PRODUCT OR ITS FITNESS FOR A PARTICULAR PURPOSE (EVEN IF EVONIK IS AWARE OF SUCH PURPOSE), OR OTHERWISE. EVONIK SHALL NOT BE RESPONSIBLE FOR CONSEQUENTIAL, INDIRECT OR INCIDENTAL DAMAGES (INCLUDING LOSS OF PROFITS) OF ANY KIND.

It is the customer’s sole responsibility to arrange for inspection and testing of all products by qualified experts. Reference to trade names used by other companies is neither a recommendation nor an endorsement of the corresponding product, and does not imply that similar products could not be used.

© 2012 Evonik Cyro LLC
All Rights Reserved.