Technical Information

ACRYLITE® Hi-Gloss

Processing Guidelines for Automotive Pillar Trims and Appliques with High Gloss NTA Acrylics

Introduction

ACRYLITE® Hi-Gloss Non Transparent Acrylics are excellent choices for exterior automotive applications based on their weatherability, toughness, gloss, and processability.

Best design practices for successful application of ACRYLITE® Hi-Gloss materials for injection molded pillar trims and appliqués involve five major areas:

- material selection
- part design
- mold design
- injection unit
- processing conditions

These five areas are interrelated and appropriate practices in each of these areas, as well as other areas which are covered in this best practice document, must be followed to ensure successful application of the use of acrylics.

1.0 ACRYLITE® Hi-Gloss Impact Modified Acrylics

A brief product description of ACRYLITE® Hi-Gloss grades are given in Table 1 below.

Material properties such as melt flow rate, specific gravity, vicat softening point, heat deflection temperature, and shrinkage are given in Table 2 to aid in the selection of grades. Technical data sheets on these grades are also available upon request.

Table 1 Product Description

ACRYLITE® Hi-Gloss FT8	High scratch resistance, standard heat acrylic
ACRYLITE® Hi-Gloss FT15	High scratch resistance, extended heat acrylic
ACRYLITE® Hi-Gloss NTA-1	Extended heat impact acrylic

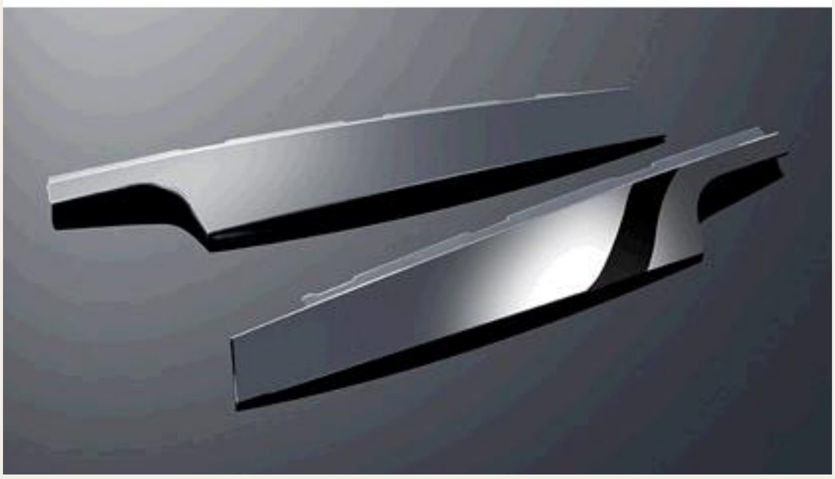


Table 2 Product Information

Material	MFR	Specific Gravity	VICAT	HDTL	CTLE	Shrinkage
	ISO 1133 g/10 min 3.8Kg / 230 °C	ISO 1183	ISO 306 °C 50 N @50°C/hr	ISO 75 °C 1.8 MPa, Annealed	ISO 11394-2 mm/mm/°C x E-5 (-30°C - 100°C) flow / cross-flow	ISO 294 mm/mm flow / cross-flow
ACRYLITE® Hi-Gloss FT8	3.5	1.187	108	101	7.56/7.25	0.006/0.006
ACRYLITE® Hi-Gloss FT15	5.3	1.192	115	105	6.68/7.20	0.005/0.006
ACRYLITE® Hi-Gloss NTA-1	3.9	1.183	110	102	9.02/9.01	0.007/0.008

2.0 Part Design

The part design criteria of major consideration for acrylic pillars and appliqués consist of appropriate attachment schemes, uniform and adequate wall thickness, generous radii around corners, sufficient draft angles, correct placement of ribs, and attention to part handling. These and other part design aspects are covered below.

2.1 Attachment Schemes

- The use of snap-fit grommets and back-side expansion slots is recommended.
- Use 1 attachment per every 4 to 6 of length of part.
- Grommets should have elastomeric washers to reduce stress and eliminate potential for squeaks.
- Self-engaging gripping clips can also be used on 2.5 mm or greater wall thickness flange sections.
- Screw attachments can be used provided there is a clip that absorbs the force from the screw threads. The screw should have a rubber gasket if it will contact the appliqué. The receiving hole in the appliqué must be 2 times the screw diameter. The screw must never put force on the appliqué wall. Torque guns should be set so as to not exceed the shear stress of the material.

2.2 Wall Thickness

- 2.5 to 3.5 mm wall thickness is recommended.
- Maintain uniform wall thickness and avoid variations that result in filling from thin to thick sections.
- Optimum thickness is often a balance between strength and weight or durability versus cost.
- Consider using features such as ribs or fillets to stiffen and add strength to parts with little increase in weight, cycle time, or cost.

2.3 Minimum Wall thickness

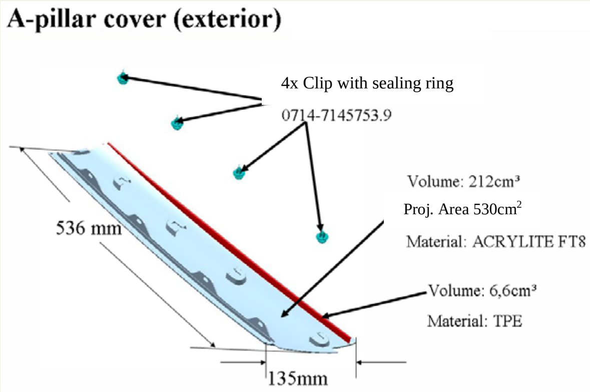
- The minimum recommended wall thickness of 2 mm (0.078 inches) allows for molding with low stress and orientation, however, for functional pillar trim, a 3.0-3.5 mm minimum wall thickness is required for suitable attachment feature strength.

2.4 Minimum Radius

- Avoid sharp inside corners on all backside features, flanges, and ribs as these corners concentrate stresses from mechanical loading, substantially reducing mechanical performance.
- The minimum recommended radius on all features is 70% of the wall thickness.
- Use generous radii at all corners.

- Ensure that the gate also has a smooth path for polymer flow, again, avoiding any sharp corners.

Picture provided by WKW. To be described by WKW.



2.5 Reinforcement Ribs

- Small 0.2 to 0.7 mm reinforcement ribs can be designed along flanges and backside features to improve part mechanical strength.
- Rib wall thickness of flange or back-side feature should be 50 to 60% of the part wall thickness.
- Above 60% could result in sink marks and read-through issues.
- Maximum rib height should be 3 times the part wall thickness.
- Intersection at the base of the rib should radii 25 to 50% of the wall thickness.
- Typical draft angle for ribs is 2° and draw polished.
- Excessive rib thickness can result in sink marks or warpage.

2.6 Locating Pins

- When using locating pins, ensure that the pin will not see cantilever forces.

2.7 Part Surface Finish

- ACRYLITE® Hi-Gloss materials will precisely replicate the mold finish and texture. Therefore, to obtain a high gloss part, the mold surface must be highly polished to a mirror finish, An SPI A-3 minimum to #1 'Diamond' or better is recommended.
- Special coatings can be used to increase the durability and polishability of the mold surface as well as improve lubricity to ease part release and removal.

2.8 Part Appearance

- Handle freshly molded parts with cotton gloves to prevent handling-related defects, otherwise, use robotics to trim.
- Use a protective film on the part immediately after trimming to prevent minor scratches. The part surface will harden as it cools.

2.9 Part Gating Location

- It is best to back-side gate, edge-gate below the beltline, or fan gate into an area that will be concealed from view.
- Locate gate at the thickest section.
- Avoid areas subject to impact or mechanical stress
- The gate must be located so the melt flows from the thicker section to the thinner section, and the number and length of weld lines are minimized and concealed from view.

2.10 Part Weld Lines

- Weld lines occur when the flowing plastic divides and later rejoins or flows together. Weld lines are normally visible as a seam and usually have reduced strength.
- Careful selection of the gate location often ensures the weld lines occur in less-visible areas of the work piece. Processing conditions are also influence weld line appearance and strength.
- Weld lines generally can be reduced by using a high mold temperature combined with a high melt temperature, a high injection pressure and the highest injection speed possible without shear-degrading the material.

2.11 Metal Inserts

- Given the proper design, inserted metal parts in injection moldings made from acrylic compounds present no major difficulties. Since plastic contracts more on cooling than metal, it shrinks onto the insert and holds it firmly.
- Often, metal parts must be secured against twisting or axial movement. In such cases, avoid designs with sharp teeth or grooves with sharp edges, because of their notch effect.
- Tensile stresses should not be higher than necessary to ensure the metal insert is seated firmly.
- Obviously, the thicker the wall of the plastic material enveloping the metal part, the less the stress will be, because forces are distributed over a larger cross-section.

2.12 Heat Staking

- Acrylic can be easily heat staked for assembly by readily using heat or ultrasonic energy.

2.13 Temporary Protective Films

- Use of a protective film on the part after molding is recommended to maintain the high gloss finish during handling and final assembly at the vehicle assembly plant.

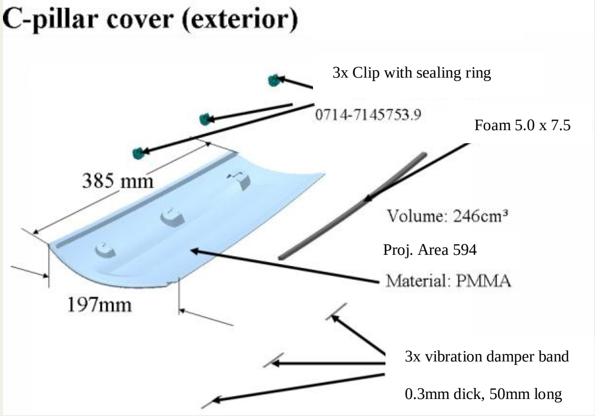
2.14 Shipping considerations

- Shipping sleeves made from foamed PP or even PE bags can be used to reduce part-to-part scratching while in transit. Felt lined shipping boxes with individual chambers are also commonly used.

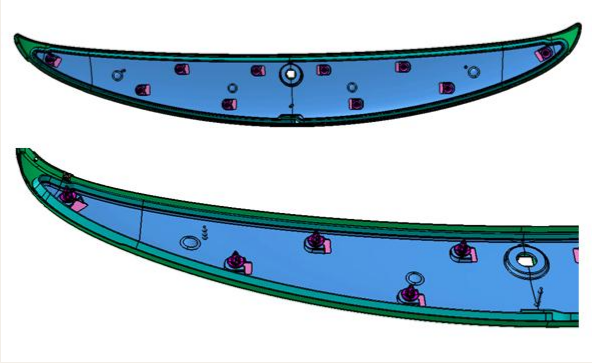
2.15 Dual Material Designs

- Elastomeric materials can be co-injection molded to produce permanently bonded flanges and weather seals. Also, acrylic can be two shot molded over ABS or PC/ABS, and PC can be molded over acrylic.

Picture provided by WKW.



Picture provided by WKW.



3.0 Mold Design

3.1 Mold Temperature Control

- The influence of the mold temperature on processing and properties of the finished part is significant. The injection into a cold mold makes it more difficult to fill the mold and may cause high cooling stresses, warping, strong orientation and sink marks in the molding.
- Cooling around gates is critical; the polymer melt must sufficiently solidify to prevent drooling. Excessive heat surrounding the gates will be exhibited as a haze, splay or artifacts of melt degradation.
- Too often the molds cooling water system cannot sufficiently cool the mold. Calculate the tons of cooling required. ACRYLITE® Hi-Gloss grades have specific heat values ranging from 2000 to 2500

J/Kg-°C, and an ejection molded-part temperature approximately 90 °C (195°F).

- Install pressure, flow and temperature indicators on the water lines. Sufficient cooling water velocity must exist to maintain turbulent flow.
- Heating and cooling in accordance with the shape of the mold and other requirements are possible with circulating-water thermostats.
- The cooling channels should be distributed so all parts of the molding cool more or less uniformly. This means that more heat can be dissipated from thick parts than from thin ones. The closer the channels are to the surface of the mold, the more effective the control of temperature.
- By separate temperature control of the two halves of the mold, it is possible to compensate for warping in the molding due to differences in wall thickness.

3.2 Sprue

- To provide enough material to fill and pack a part with a cold sprue, the Sprue O-diameter (small end of the sprue), should be at least 1 ½ times larger than the part's nominal wall thickness.
- A heated sprue bushing orifice diameter should be at least 3/4, and as much as 1 ½ times the part nominal wall thickness.
- The Sprue O-diameter should be 0.050 in. (1.27 mm) larger than the main runner diameter.
- The taper of the sprue is normally 3˚.
- The sprue must have a polished surface to ensure low resistance during ejection.
- In the cold well, an ejector pin with an undercut or Z-puller can be fitted to pull the sprue from the busing. This is suitable for any two plate mold with a runner system.

3.3 Runners

- Full round runners are best. Oval and rectangular cross-sections are not recommended.
- Keep the runners as short as possible.
- Successful designs usually start with a runner diameter feeding the gate at least 1 ½ times the thickest section of the part. Main runner diameters are often 0.250 in. (6.35 mm). The melt is not highly oriented in the runners; therefore the pressure drop in the runners can be much greater than normally anticipated.
- Venting on the runners should be 0.002 in. (0.051mm) deep, as wide as the runner, with a 0.060 in. (1.52 mm) land, dropping into a 0.040 in. (1.02 mm) channel to atmosphere. Polish the vent lands to an A1 finish to make them self-cleaning.
- The gate must be centered on the runners to eliminate excessive shear.

3.4 Gates

Fan and Tab gates

- Fan and tab gates are the most successful designs for ACRYLITE® Hi-Gloss.
- Fan and tab gate depth must be a minimum of 75% of the wall thickness, and the width about 10 to 40 times the gate depth.
- The gate depth controls freeze off time.
- The gate land must be as short as possible, 0.030, 0.050 in., and never exceed 0.050 in (1.27mm).
- Rounded edges, where the gate meets the part, reduce the tendency to jet.

Valve Gates

- Valve gates with low shear designs and proximity cooling have also been successful. Do not ignore the pressure drop in the annular flow around the valve gate pins. The pressure drop may be sufficiently high due to the annular flow width and length that ACRYLITE® Hi-Gloss may not be successfully molded.

Submarine Gate

- With a submarine gate, the molding is automatically separated from the runner as it is removed from the mold.
- Particularly with hard and rigid acrylic, moldings must be removed as early as possible from a mold with a submarine gate. This is so the plastic in the gate channel can be removed from the tunnel without breaking off the runner.

3.5 Draft angle

- Draft angles are necessary for easier removal of the part from the mold.
- 5° is recommended where possible.
- The minimum draft angle is 1°.
- Highly polished, short cores may allow a 1/2° draft.
- Textured surfaces require 2° or more draft.

3.6 Vents

- Vents must be 0.002 in. (0.051 mm) deep, 0.200 in. (5.1mm) wide, with a 0.040 in. (1.02 mm) land, dropping into a 0.040 in. (1.02 mm) channel to atmosphere.
- Draw polish the vent lands to help make them self-cleaning.

3.7 Undercuts

- With a standard mold, it is normally impossible to produce acrylic parts with undercuts, since they would break on removal. Slide molds are, therefore, necessary.

3.8 Notch Effects

- Like all amorphous polymers, acrylics are relatively hard, rigid, and sensitive to notches.
- Transitions in the shape involving sharp edges must be avoided.
- Even small radii at such transitions greatly increase the fracture resistance.
- Since places such as gates, especially restricted gates, behave as notches, they should be located at some less-stressed part of the molding.

3.9 Maximum shear stress and shear rate

- The mold's combined sprue, runner, gate and wall dimensions must be sized to allow filling at medium injection speeds with shear rates below 40,000 1/s and shear stresses below 58 psi (4 bar).

3.10 Mold Shrinkage

- Shrinkage for ACRYLITE® Hi-Gloss materials are given in Table 2.
- Shrinkage is mostly influenced by part configuration and processing conditions.
- Shrinkage is also higher further from the gate.

3.11 Mold Surface finish

- The mold cavity must have a completely smooth and nonporous surface. Case hardened, highly polished steel or chrome plated surfaces produce glossy appearing parts.
- An SPI A-3 minimum to #1 Diamond, or better finish is recommended for high gloss moldings.

3.12 Mold Cavity Maintenance

- The mold cavity must be kept from getting scratched during routine cleanings. Mold cleaning should only be done by qualified personnel.
- Soft, non-abrasive cleaning agents and materials should be used.
- Typical hard-chrome surfaces can last up to several years of continuous use without refinishing.

4.0 Injection Unit Requirements

4.1 Injection Unit

- The Injection molding machine must be capable of generating sufficient clamp force.
- To maintain reasonable residence times and minimize shear degradation, the shot size should range from 40 to 60% of the barrel capacity.
- If the cylinder is too large, difficulties in processing may occur because of long

residence times or because of excessive stress on the machine drive.

4.2 Mold Clamping Unit

- The mold clamp is generally stressed the most during the production of components with large surface areas.
- ACRYLITE® Hi-Gloss requires 2.5 tons/in^2 (352 kg/cm^2) of projected area for flow length/wall thickness (L/t) <100/1 and 5 tons/in^2 (703 kg/cm^2) for L/t >100.
- To estimate the necessary clamping force, assume a pressure in the mold of between 3,800-4,500 psi on the projected surface.

4.3 Screw Design

- The screw should be conventional design, three sections with a non-return valve.
- A chrome plated screw reduces deposits on the screw's surface thereby reducing black specs in the molded parts.
- Ideally, the shot size should be 40-60% of the cylinder capacity.
- Channel depth compression ratios may range from 2.0:1 to 2:8:1. The problem of overheating is more related to channel depth and shear rather than compression ratio.
- The feed zone length should be 60% of the screw's length and the compression zone should be 20% of the screw's length.
- Successful custom screws with a feed section as low as 45% and a 35% transition section have been observed in the field.

4.4 Nozzle and Tip

- ACRYLITE® Hi-Gloss require a full taper nozzle on the molding machine to eliminate dead spots and shear and degradation points.
- The nozzle orifice must be no more than 10% smaller than a cold sprue's O-diameter (small end of the sprue) and the nozzle's radius must match the sprue bushing's radius.
- The nozzle must match the flow tube diameter of a hot runner manifold. Long nozzles must be uniformly heated and accurately controlled.

5.0 Processing Conditions

5.1 Conveying

- The high surface hardness of acrylic materials can be abrasive in conveying systems.
- Stainless steel should be used for fixed conveying lines.
- Polyurethane hoses can be used for short flexible hose runs.
- PVC flexible hoses must never be used as it softens and feeds PVC particles into the feed stream.

5.2 Drying

- ACRYLITE® Hi-Gloss absorb some moisture from the atmosphere.
- The shipping specification is 0.15% maximum moisture. An open box in a hot and humid atmosphere can absorb up to 1.5% moisture.
- For consistent, quality processing, moisture content should be 0.08% or less.
- Drying ACRYLITE® Hi-Gloss at 175 °F (80° C) for three hours in a desiccant dryer with the effluent air set to a -20°F (-29 °C) or lower dew point is sufficient in most cases.
- When material is suspected to contain high levels of moisture, four to six hours of drying may be required.
- The plastic in the dryer should move in a plug-flow fashion, so that all material is dried for essentially the same length of time.
- Failure to dry adequately can result in a slight reduction in surface gloss, to severe surfaces streaks and/or bubbles. To minimize moisture absorption, containers should be sealed when not in use.

5.3 Purging

- Purging must be performed to prevent poor part appearance.
- Many materials such as styrene, styrene-based materials and even polycarbonate cannot be purged out by ACRYLITE® Hi-Gloss.
- In most cases, un-dried clear ACRYLITE® acrylic, H15 grade is a sufficient purging compound. Commercial compounds such as ASAClean, Dynapurge and Ultimax are recommended.

5.4 Barrel Temperatures

- ACRYLITE® Hi-Gloss are best molded with a low melt temperature and moderate to high injection speed.
- Barrel heats should be profiled from 380 °F (193 °C) at the feed section to 445 °F (230 °C) in the compression section to obtain a melt temperature of 440 to 450 °F (225-232°C).
- Please see the specific product processing sheets for additional grade details.
- In some situations, the best temperature may be outside of the ranges indicated. For example, if the residence time in the cylinder is usually long (short), the cylinder temperatures must be set lower (higher).
- ACRYLITE® Hi-Gloss compounds have good thermal stability, so that temporary interruptions in production usually do not cause degradation problems.
- Prolonged interruptions may lead to slight degradation of the plastic in the cylinder, so some purging may be required.
- Higher melt temperatures will not necessarily correct poor filling due to low injection pressure, excessively thin walls, or undersized sprue, runners or gates.
- The properties of the finished part can be influenced by the melt temperatures. As melt temperature increases, molecular orientation and stresses are reduced while the possibility of sink marks increases. It is best to work at the lowest temperature at which sinks do not occur.
- For very thick parts, the temperature may be at the lowest end of the recommended range.

5.5 Nozzle Temperature

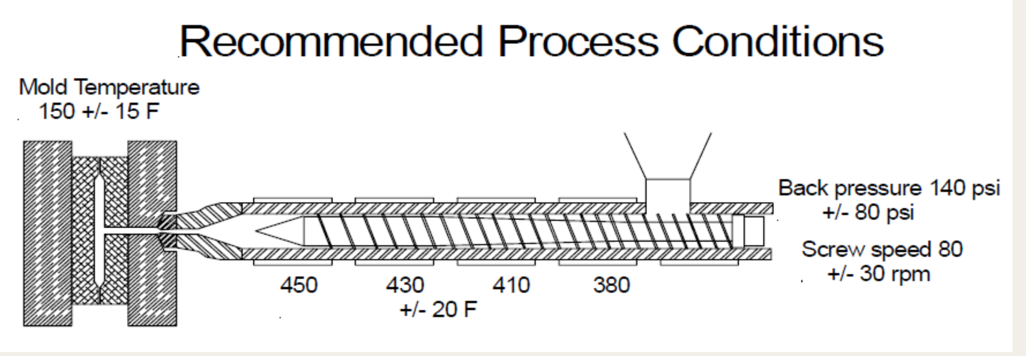
- Incorrect heating of the nozzle may result in various processing faults.
- When the cycle is short, the nozzle tends to run hotter. The nozzle temperature should, therefore, be set a little below the nearest cylinder zone to avoid the effects of overheating.
- When the cycle is long, the nozzle is in longer contact with the relatively cool mold so that it dissipates heat. This may lead to cold marks. These are avoided by having a relatively high nozzle temperature.

5.6 Mold temperatures

- ACRYLITE® Hi-Gloss have the lowest stress and highest gloss with a 150 °F +/- 15 °F (65 °C +/- 9°C) mold surface temperature.
- Ensure sufficient, turbulent cooling water flow with approximately 35psi (2.4 bar) pressure drop across the mold, and no more than 5°F (3°C) heat differential between the waterline's inlet and outlet.

5.7 Hot manifolds, runners and gates

- Due to minimal decomposition and a wide working temperature range, ACRYLITE® Hi-Gloss compounds are readily worked in hot-manifold molds. In these molds, the compound is maintained at the processing temperature by heaters on its way from the cylinder nozzle to the gates.



- The plastic remains molten in a runner plate with hot manifolds. Heat is usually generated by resistance fitted into the runner plates. The runner plate should only contact other parts of the mold at a few places in order to keep down heat transfer to the mold.
- Hot runners must be appropriately sized by the hot runner supplier.
- Undersized hot runners produce unacceptably high pressure drops, resulting in poor parts and poor yields.
- Hot runner designers must use the exact shear viscosity data and recommended processing conditions available from Moldflow® Corporation.
- Moldflow®'s shear viscosity data for specific grades are available upon request.
- To prevent build-up of degraded material within the hot manifold & runner systems, institute strict procedures requiring immediate reduction in manifold and runner temperatures during any process upsets lasting longer than 2 minutes.

5.8 Screw Speed

- A starting point for screw speed is 80 +/- 30 rpm.
- With a large cylinder and complete utilization of the shot, the speed can be slower. When cycles are short, higher screw speeds are often necessary in order not to lengthen the cycle through plasticizing time.

5.9 Back Pressure

- During the return of the screw, back pressure avoids air inclusion in the solids feed section. If air is present, it can be seen at irregular intervals in an air short.
- The temperature of the compound and the mixing action increases with rising back pressure, while screw recovery time is increased.
- Back pressure can vary from 50 to 250 psi (3.5 to 17 Bar) depending on the screw design, capacity, and wear.

5.10 Filling and Injection Speed

- The ease of mold filling will depend on the wall thickness and maximum distance the plastic must flow from the sprue, that is, the flow path.
- Fill the cavities to 98% full at a medium to fast injection speed to maintain constant viscosity.
- Mold and gate designs should allow fast filling speeds.
- Injection that is too fast often results in haze, splay marks and sink marks.
- Some mold designs have required extremely slow filling speeds. Slow speeds result in difficult filling and packing control.

5.11 Injection Pressure

- The injection pressure largely depends on the shape of the mold, and the pressure loss between the cylinder and the cavity.

- The pressure loss can be kept low by means of short, sufficiently-large sprues and runners in the mold, and an ample nozzle opening.

Here are typical values for the initial injection pressure settings:

Molding Machine Type: Screw Injections

Thick-walled and Thin-walled articles:
6,000-15,000 psi up to 18,000 psi

Thick-walled articles with long flow paths:
Up to 18,000 psi

5.12 Holding Pressure

- Holding or packing pressures should be less than the filling pressures to ensure a low stress gate area.
- Holding pressure also affects the properties of the molding. If substantial internal stresses must be prevented, the level and duration of the holding pressure need only be enough to produce a satisfactory molding.
- A high and long holding pressure avoids conspicuous sink marks, particularly on thick-walled molding. Smaller holding pressure may be advantageous for complex articles.

5.13 Handling after ejection

- Ejected parts will have a static charge possibly attracting surrounding dust and dirt.
- Clean conveyors, cooling stations and floors before initiating full production. Handle parts with gloves until fully cooled.
- Package parts with chemically compatible materials; that is, do not use plasticized plastic wraps. Wraps should be tested for compatibility.

5.14 Regrind

- ACRYLITE® Hi-Gloss can be reground and reprocessed without adversely affecting physical properties.
- A concentration up to 25% is recommended.
- It is important to avoid contamination and remove fines in the regrinding process. Evonik Cyro recommends the use of Pelletron Corporations Dedusting units.
- Due to increased surface to volume ratio, the regrind may require additional drying.

6.0 FINISHING AND POST TREATMENT

6.1 Separation of the Sprue

- To avoid having to perform another operation, the sprue is best taken off immediately after removal from the mold.
- Thin gates are cut with heavy duty gate cutters, heated diagonal cutting pliers or are broken off. Breaking off can cause problems. If there is strong orientation in the gate area, the molding may splinter or tear.
- Tab gates are usually removed with a small circular saw, while sprues are removed with cutters. The short, residual stump may be faced on a milling machine.
- Sprues can also be successfully removed by laser degating. This produces a very clean cut, but some stress is introduced into the part.

6.2 Machining

- Moldings made from acrylic compounds can be machined. A minimum of heat should be generated at the machining point in order to avoid smudging. This is obtained with correct tools, and a cutting rate that is not too fast.
- Heat can also be dissipated with coolants such as drilling emulsions or water. These liquids are placed on the tool, just as they are for metalworking. Crazing agents such as alcohol must not be used for cooling.

- Fairly severe local heating produces finishing stresses, and must be avoided.
- If an improperly finished molding is attacked by a stress cracking agent, cracks will form at the edges of drilled holes.

6.3 Antistatic Treatment

- Use ionized air on conveying belts.
- The attraction of dust can be prevented by surface treatment with liquid antistatic agents.
- Moldings can be immersed in this liquid immediately after removal from the mold.

6.4 Annealing

- Annealing is recommended to avoid stress crazing if the molding may contact solvents or swelling agents.
- Black pillar and appliqué parts may anneal themselves upon exposure to the sun, thus relieving any molded-in stress.
- Moldings that will be bonded or painted may require annealing. Internal stresses are reduced by annealing at the highest possible temperature, although the heat distortion temperature of the molding must not be exceeded.
- Below is a procedure for annealing:

1. Preheat oven to 180 °F (82 °C) or ~15 °F below the material's DTL. Place parts into oven. Heat parts in the oven for 1 hour per mm of thickness.

Examples:
1 ½ hours heating for 1.5 mm (0.060.) thick parts
2 hours heating for 2mm (0.079.) thick parts
2 ½ hours heating for 2.5 mm (0.098.) thick parts

2. The next step is the most critical. The parts must cool very slowly. Do not cool the parts faster than 40°F (20°C) per hour. Many processors simply turn the oven heat off and let parts cool slowly. The parts are considered sufficiently cooled when they have reached ~104 °F (40 °C) or below.
3. Cooling time should be equal to or longer than 1 hour per mm of thickness.

Example:
1 ½ hours cooling for 1.5 mm (0.060") thick parts
2 hours cooling for 2mm (0.079") thick parts
2 ½ hours cooling for 2.5 mm (0.098") thick parts

Possible variations:
Heating time may be reduced if you only wish to anneal stresses on the part's outer surface. Half the recommended heating and cooling times have been found to be sufficient. You must test if less annealing is sufficient for your application.

Heating temperature and times may be varied, but you must ensure the parts are not overheated and possibly deformed. Parts will deform when heated above the DTL. Heating a part at 190 °F for 30 minutes and then slowly cooling at the recommended rate is possible but must be tested.

6.5 Stress Test

- The stress test consists of an ethyl acetate test which is a relatively simple and informative way to determine molded-in stresses. It can also be used in production control.
- The molding is immersed in ethyl acetate (98% pure) for 60 sec, both molded part and the ethyl acetate should be at room temperature (72°F).
- The molding is dried in air for five minutes and then examined for cracks and crazing.
- Properly molded parts or an annealed part will not craze and will, as a matter of experience, meet all practical requirements.
- You can also immerse the parts for different intervals and longer periods of time to determine relative stress levels.

6.6 Bonding

- There are a number of methods for bonding ACRYLITE® Hi-Gloss compound

to acrylics or other materials. Solvents such as methylene chloride can be used, as can polymerizing adhesives. There are adhesives with exceptional bond strength, good body and very good weather resistance such as cyanoacrylates.
- Because of the internal stresses that occasionally occur in injection-molded parts, crazing may occur as the result of solvents used during bonding. In order to avoid this, the molding should be annealed before bonding.
- The possibility of subsequent crazing is reduced if the design of the joint allows the solvent to escape.

6.7 Stress Cracking

- Molded components may contain inherent stresses due to processing or exposure to stresses in use. Even if the stresses do not exceed the breakage strength of the plastic, cracks may be formed under the influence of a solvent or swelling agent (stress cracking).
- Following the proper molding and design guidelines and using the proper attachment scheme will yield a part with low molded-in and assembly stress.
- If molded components will contact known substances whose effect is not known, stress-cracking behavior should be tested beforehand.

7.0 Polishing of Pillar and Appliqué Parts Made from Black ACRYLITE® Hi-Gloss Molding Compounds

PMMA is one of the hardest uncoated plastics, meaning that it does not scratch as easily as softer materials. Any scratching that does occur, which is particularly noticeable on high-gloss black surfaces, can be removed by polishing as described below.

Minor scratches can be removed by polishing with commercially available car polish. This is best done using a soft fabric, according to the manufacturer's instructions. Polishes should be tested first to determine compatibility with acrylic. Do not polish in direct sunlight. Polish in a covered, shaded surrounding only.

Deep scratches can be removed using 3MTM Finesse-it™ Finishing & Polishing System Kit and following the manufacturer's instructions.

Agents used:
- 3M™ Finesse-it™ Finishing & Polishing System
- 3M / Superabrasive and Microfinishing Systems Department

3M Center, Bldg. 251-52-04
St. Paul, MN 55144-1000
651-737-1783

Example of Polished Pillar
Panel was scratched along its entire length.

2/3 of the surface was treated with wet abrasive paper (still visible on middle section)

The upper third of the surface was polished to a high gloss.

8.0 Chemical Resistance

8.1 Chemical Resistance

- ACRYLITE® Hi-Gloss compounds resist most chemicals that are found in normal use.
- For your reference a table of chemical resistance is attached. Scratched surface

scratches remove by sanding surface polished to high-gloss finish
- Because of their chemical structure, ACRYLITE® Hi-Gloss compounds are dissolved by most organic solvents, such as aromatic hydrocarbons; however, they do resist aliphatic hydrocarbons.
- Observe care when ACRYLITE® Hi-Gloss compounds are used with plasticized thermoplastic materials and elastomers. Some plasticizers may migrate, especially at elevated temperatures, and then attack ACRYLITE® Hi-Gloss compounds.

8.2 Chemical stability of injection molded ACRYLITE® Hi-Gloss compounds

- These data were developed at a test temperature of 68°F (20°C), and a relative humidity of 50%. Molded components were exposed to low stresses.
- In practice, resistance is dependent not only on internal and external stresses, but also to a large extent on orientation in the molded component.
- In addition, the tendency to dissolve and swell is altered considerably with the temperature. As a result, we recommend that appropriate tests should be carried out in doubtful cases and technical advice requested from Evonik Cyro.

+ Resistant   x Limited   – Susceptible

Drinks and edible liquids
+ Beer, wine, fruit juices
x Coffee, tea
x Liqueurs, see alcohol
+ Milk, chocolate
+ Cooking oil
+ Vinegar
+ Water, mineral water

Spices
+ Aniseed, bay leaves, nutmeg
+ Pepper, cinnamon, onions
– Cloves

General
+ Photographic baths
– Nail polish

Inorganic substances
x Chromic acid
+ Hydrofluoric acid, up to 20%
+ Phosphoric acid, up to 10%
+ Nitric acid, up to 20%
x Nitric acid, 20 to 70%
– Hydrochloric acid
+ Sulfuric acid, up to 30%
+ Sulfuric acid, up to 5%
x Sulfurous acid, concentrated
– Sulfur dioxide, liquid

Greases and oils without additives
+ Animal
+ Mineral
x Vegetable

Paints, waxes, etc.
x Wax polish
– Cellulose paints
– Paint thinners
+ Pure-oil paints l
x Acrylic paints

Gases
+ Ammonia
x Bromine
x Chlorine
+ Carbon monoxide
+ Carbon dioxide
+ Natural gas
+ Methane
+ Ozone
+ Oxygen
+ Sulfur dioxide (dry)
+ Nitrogen dioxide
+ Nitrogen monoxide

Organic solvents and plasticizers
– Acetone
– Phenols
– Ether
+ Propylene
– Ethyl acetate
– Pryidine

+ Ethyl alcohol, up to 15%
– Carbon disulfide
– Ethyl alcohol, concentrated
+ Diethylene glycol
– Ethyl bromide
– Ethyl butyrate
– Ethylene bromide
– Amyl acetate
– Aniline
– Benzaldehyde
– Sulfur dioxide, liquid
+ Calcium hypochlorite
– Benzene
x Butanol
– Methyl chloride
– Chlorinated hydrocarbons
– Chlorophenol
+ Cyclohexane
– Diacetone alcohol
X Diamyl phthalate
+ Turpentine
+ Tricresyl phosphate
– Xylene
+ Diethylene glycol
– Dibutyl phthalate
– Dioxane

Disinfectants

+ Bleaching powder solution, up to 2%
+ Bleaching powder paste
– Tincture of iodine, 5%
– Carbolic acid
+ Hydrogen peroxide, up to 40%

Alkalis

+ Caustic potash
+ Whitewash
+ Soap suds
+ Soda
x Glycol
+ Heptane
+ Hexane
x Isopropyl alcohol
– Cresol
– Methyl ethyl ketone
x Methanol, up to 15%
– Methanol, concentrated
– Lactic acid butyl ester
x Perchlorethylene
+ Petroleum ether
x Paraffin

9.0 Solving Common Molding Problems

Various defects may occur in injection molding. It is often difficult to quickly determine the real cause of a given problem. A listing of problems and remedies that can be tried systematically follows.

1. Bubbles inside the molded articles.
a) Increase the mold temperature
b) Reduce the compound temperature by lowering the temperature of the heating band, the screw speed and the back pressure
c) Extend the injection cycle
d) Increase the holding pressure
e) Heat the rear end of the cylinder more intensely, reduce or turn off the water cooling for the cylinder
f) Reduce injection speed
g) Extend the post pressure period
h) Increase the runners and the nozzle diameter
i) Heat the runners

2. Articles not fully molded (short shot).
a) Increase the compound temperature by raising the heat band temperature, the screw speed and the back pressure
b) The cylinder has too much resistance and must be altered or replaced
c) The cross-section of the shut-off type nozzle is too small; if necessary use an open nozzle
d) Heat the rear cylinder heating band to a higher temperature
e) Make sure that the feed setting is adequate
f) Avoid excessive supply of material
g) Increase the gate's size, the runner and the nozzle diameter. Avoid flat runners
h) Increase the tool temperature
i) Increase the injection pressure
j) Use a larger machine
k) Reduce the injection cycle in the case of plunger machines, otherwise the plasticizing capacity will not be sufficient
l) Increase the injection speed

3. Surface imperfections such as flow lines and spray marks.
a) Pre-dry at a higher temperature and extend the pre-drying time.
b) Reduce the compound temperature by lowering the heating band temperature, together with the screw speed and back pressure
c) Increase the tool temperature
d) Increase the injection pressure
e) Increase the rate of feed of the plunger or screw
f) Enlarge gates, runners or nozzle top ensure that the mold is filled more quickly

4. Sink marks.
a) Increase the mold temperature
b) Reduce the compound temperature by decreasing the heating band temperature, the screw speed and the back pressure
c) Increase the holding pressure
d) Lengthen the period of holding pressure
e) Reduce injection speed
f) Increase the temperature at the rear end of the cylinder
g) If plasticizing capacity of the machine s too low, lengthen cycle timer
h) Use larger gate and sprue channels in order to prevent premature setting
i) Increase the fee
j) Place molded articles in lukewarm water immediately after removal from the mold

5. Sprue breakage.
a) Make sure the sprue bushing is conical and larger than the nozzle
b) Make sure that the nozzle is correctly seated and has the correct shape
c) Increase undercutting at the extractor
d) Round off Z shaped extractors slightly, to prevent breakage of the sprue at this point
e) Polish the sprue bushing
f) Reduce the injection pressure
g) Reduce the holding pressure period
h) Reduce the injection pressure. If the sprue is still too soft, cool the mold in the vicinity of the sprue bushing or use a narrow sprue system

6. Air bubbles at the edge.
a) Reduce injection speed
b) Reduce the cylinder temperature
c) Provide adequate mold venting
d) Alter the wall thicknesses of the molding

7. Deformation of the moldings after ejection.
a) Increase the cooling time
b) Heat the two halves of the mold separately; cool hollow sides, keep the opposite side warm
c) Arrange tempering channels just below the mold surface
d) Reduce the compound temperature by lowering the temperature of the heating band, as well as the screw speed and the back pressure
e) Reduce the tool temperature
f) Use a device to clamp the moldings.

8. Weld lines.
a) Increase the mold temperature
b) Increase the compound temperature by raising the heating band temperature, the screw speed and the back pressure
c) Increase the injection pressure
d) Increase the injection speed
e) Change location of gate
f) Alter the wall thickness at given points of the molding, in order to obtain a uniform flow of material when the mold is filled
g) Provide better venting for the mold

9. Creases and dull spots across the direction of flow (especially in the case of thick components).
a) Reduce injection speed
b) Increase the mold temperature
c) Increase the compound and the nozzle temperature
d) Round off gates and polish well
e) Provide cold slug well

10. Weld lines behind inserted recesses and letters.
a) Increase the mold temperature
b) Reduce the compound temperature by lowering the heating band temperature, the screw speed and the back pressure
c) Increase the injection pressure

d) Reduce injection speed
e) Change location of gate
f) Reduce the depth of the lettering
h) In certain cases an increase in the compound temperature may help

11. The molding breaks on ejection.
a) Check the tool for undercutting
b) Improve the polish of the tool
c) Fit additional ejectors
d) Reduce the injection pressure
e) Reduce the tool temperature
f) Do not cool the part excessively

12. Streaking inside the molding.
a) Do not mix different ACRYLITE® acrylic molding compounds
b) Do not add ground sprue to fresh material
c) Reduce the injection speed
d) Increase the sprue diameter and provide rounded edges on gates
e) Increase the compound temperature by increasing the heating band temperature, the screw speed
and the back pressure
f) Increase the nozzle temperature
g) Reduce the speed of the screw

13. The strength of the molding is inadequate.
a) Reduce the cylinder temperature in the case of long articles.
b) Increase the cylinder temperature for flat articles

c) Arrange the gate at a point which is not subject to high stress
d) Use a tab gate
e) Increase the wall thickness
f) Avoid sharp transitions in the molding

10.0 Purge Compounds and Dedusting Equipment Suppliers

Purging compounds
Ultimax 9100
Claude Bamberger Molding
Compounds Corp.
111 Paterson Plank Road
Carlstadt, NJ 07072
201 933-6262
www.Claudebamberger.com

ASA Clean .Sun Plastech
1140 Parsippany, NJ 07054
800 787-4348
www.asaclean.com

Dynapurge "C" for clear resins
Shuman Plastics
35 Neoga St.
Depew, NY 14043
716-685-2121
www.shuman-plastics.com

Dedusting equipment
Pelletron Corporation
499 Running Pump Rd
Lancaster, Pa 17601
Tel: 717-293-4008
fax: 717-293-4011
www.pelletroncorp.com

Evonik Cyro LLC 299 Jefferson Road, Parsippany, NJ 07054-0677 USA
Phone: 800-631-5384; Email: cyro.polymer@evonik.com; Website: http://www.acrylite-polymers.com
Technical Support: visit the TechKnowlogy Center at cyro.custhelp.com