



Injection Molding Compound Processing Guide

INTRODUCTION

Geon developed this guide to assist in the successful injection molding of Fiberloc® composites. It includes recommendations for equipment, mold and processing, plus a quick reference trouble shooting guide. As with the processing of any new material, we suggest you work with a Geon technical service specialist on your initial run of Fiberloc composites. Our highly trained technical service team is also available whenever you may need processing assistance.

You'll find that Fiberloc composites possess processability similar to vinyl (PVC,) but parts molded from Fiberloc composites exhibit enhanced strength, stiffness and dimensional stability in addition to vinyl's typical properties. That's because Geon's chemical coupling technology provides a strong bond between high flow, engineering vinyl polymers and glass fiber reinforcement.

This bond gives Fiberloc composites the unique balance of properties that make Fiberloc an ideal replacement material in applications previously made from metal or traditional engineering thermoplastics.

- High strength and stiffness
- Dimensional stability
- Creep resistance
- Low thermal expansion
- Fatigue resistance
- Low mold shrinkage
- Chemical and corrosion resistance
- Flame resistance
- Low water absorption UV stability
- Processability
- Colorability

Geon bulletin CIM-05 (Design Guide)

provides specific property data for Fiberloc injection molding compounds.

EQUIPMENT

Machine Type

Fiberloc compounds should be molded only on properly equipped reciprocating screw injection molding machines. Because vinyl is mainly an amorphous material (no sharp melting point), a plasticating screw is required to prepare a homogeneous melt for injection into the mold cavity. Plunger/ram machines are not suitable for Fiberloc compounds, due to poor mixing, material stagnation and resulting degradation.

Machine Size

Clamp Capacity

A machine having a minimum clamp force of 300 to 400 kg/cm² (2 to 3 tons/square inch) of projected part area, including runners, is recommended.

Barrel Capacity

To obtain the widest processing latitude and optimum physical properties of Fiberloc polymer composites, an appropriate match of shot size, i.e., the volume of cavities, runners and sprue, to the barrel capacity is desirable. A shot weight of 60% to 75% of barrel capacity (rated in the Fiberloc material) is recommended. This shot size minimizes melt residence time in the barrel, thus enabling processing at higher melt temperatures and with optimum melt flow, while at the same time, avoiding degradation. Since the optimum match of barrel capacity is not always practical due to clamp requirements or machine availability, shot size as low as 30 to 35% may be used with the understanding that

the processing latitude of the material may be reduced, and that as a result, the ultimate physical properties of the material may not be fully developed. When utilizing the lesser barrel capacities, lower melt temperatures may be required to prevent thermal degradation due to the longer residence time in the barrel. Lower melt temperatures mean higher melt viscosity and more resistance to flow. Greater injection pressures will be needed to fill the part which may produce molded-in stresses that could adversely affect dimensional stability and other properties of the finished part. Higher utilization of barrel capacity is therefore recommended to reduce residence time.

When calculating optimum barrel usage for Fiberloc composites always consider the specific gravity of the material versus the specific gravity of the material for which the machine was rated. Most machines are normally rated in ounces of general purpose polystyrene. Example: Using a specific case of Fiberloc 80520 (specific gravity = 1.47 from data sheets) and given that general purpose polystyrene has a specific gravity of 1.05, a 1.7 kg (60 oz.) barrel rated in general purpose polystyrene will deliver 2.3 kg (82 oz.)* of Fiberloc 80520.

$$*1.7 \text{ kg} \times \frac{1.43}{1.05} = 2.3 \text{ kg or}$$

$$60 \text{ oz.} \times \frac{1.43}{1.05} = 84 \text{ oz.}$$

A recommended shot weight, including sprue, runners and parts would then be 1.7 kg (62 oz.) on this machine. (2.3 kg x 75% of capacity = 1.75 kg or 84 oz. x 75% of capacity = 63 oz.). The size should not fall below 35% capacity (0.81 kg [29 oz.]). Correct specific gravities for calculating the desirable shot size ranges from other Fiberloc compounds can be obtained from compound data sheets.

Barrels

Injection molding machine barrels should be constructed of wear resistant materials for long term processing of glass reinforced materials. The most common method used to protect the barrel is deep nitriding which gives adequate protection for occasional molding glass reinforced compounds.

For long runs two methods known to give superior wear resistance are Bimetallic runnings and fully hardened tool steel inserted linings. The preferred Bimetallics should be a tungsten carbide in a nickel alloy matrix. The most wear resistant tool steels are high sinered powered metal high vanadium tool steels. A stainless grade T440V* or T420 V is the preferred material for these inserts. This steel gives excellent wear resistance and good corrosion protection.

Care should be taken to assure that the materials of construction of the screw and barrel are compatible so as to avoid adhesive wear caused by galling. Most screw and barrel manufacturers can supply recommendations of appropriate screw/barrel materials combinations.

*Trademark of Crucible Specialty Metal Crucible Particle Metallurgy.

Screw Design

Screws having compression ratios of 2.0 - 3.0 and length to diameter ratios of 16/1 to 24/1 are recommended for processing Fiberloc compounds. Higher compression ratio screws should be avoided because of increased and excessive shear heating. Fiber breakage and loss of physical properties are also possible because of the increased abrasion and wear on screws and barrels.

Most injection molding machines come equipped with flame hardened 4140 steel screws as standard equipment. While these screws are adequate for non- reinforced plastics, *they are not suitable for long term production of abrasive materials.*

A properly designed screw made of corrosion and abrasion resistant materials is essential for economical long-term production of Fiberloc. Materials systems used for these screws are:

Fully hardened tool steel (D2, h13)
(protected by a corrosion abrassive resistant coating)

(Xaloy 830)¹ UCAR LW-1N302, NYE-CARB³, JET KOTE4, (various nitride) or thick build-up coats of hard Chrome or Fully hardened T420V or T440V stainless tool steel.

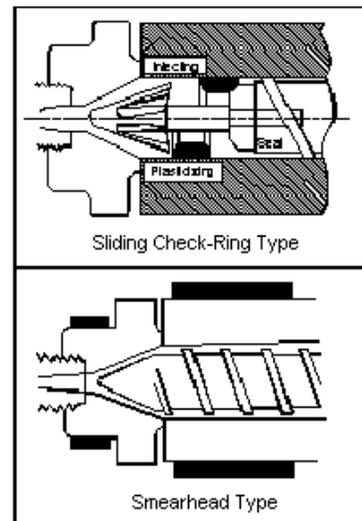
Most molding machine manufacturers offer these type screws as an option for new machines and as replacement parts. These type screws are also available from many screw manufacturers and rebuilders.

1. Trademark of Xaloy Corporation
2. Trademark of Union Carbide Corp.
3. Trademark of Electro-Coatings. Inc.
4. Trademark of Cabot Corporation.

Screw Tip

A *free flaw design check ring* made of tool steel protected by an abrasive and corrosion resistive coating (as described above for the screws) or T-440V or T-420V is the preferred tip for processing Fiberloc molding compounds. These tips are available from many molding equipment suppliers and molding machine manufacturers.

Figure 2
Screw Tip Design



Smear head tips can be used for some applications, though shot control problems and packing deficiencies limit their utility. Ball check tips and restrictive check rings should not be used as they can cause stagnation or high shear which could lead to problems in material decomposition and excessive glass breakage.

EQUIPMENT

BARREL END CAP AND NOZZLE

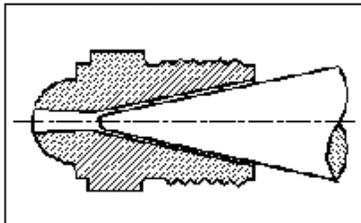
The barrel end cap should taper smoothly from the barrel diameter to the nozzle rear opening as shown in Figure 3.

The nozzle length should be as short as possible, and the nozzle should be equipped with a separate heater control. A provision for thermocouple monitoring of the nozzle temperature is necessary. The thermocouple should not project into the melt stream. Proportional, solid state, temperature controllers are also strongly recommended. Depending on temperature requirements, a silicon controlled rectifier (SCR) or triac thyristor circuit may be used. The usual variac or on/off relay controls are not as effective for maintaining the processing control desired for Fiberloc compounds.

The Full Internal Taper nozzle design is preferred. Ideally, shear effects on the material are minimized by "zero" land link of nozzle orifice diameter.

The nozzle discharge-orifice diameter should be at least 0.6 cm (0.25 in.).

Figure 4
Optimum Nozzle Design

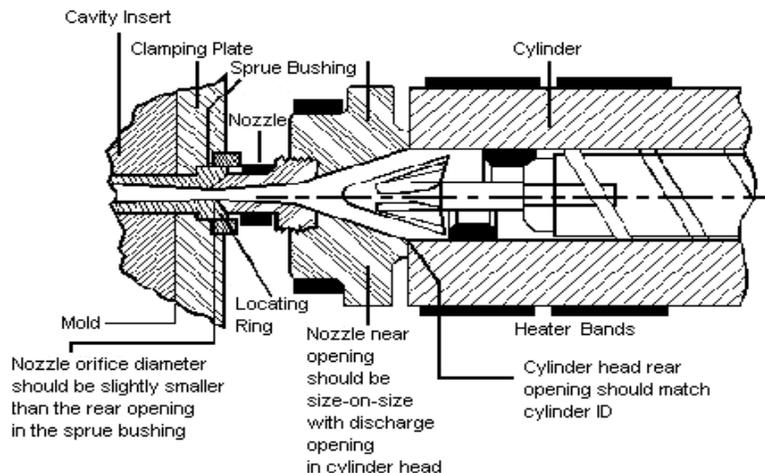


Optimum Nozzle Design

Small orifice nozzles restrict melt flow. Short shots, sink marks, shear bums or other defects may occur. In all cases, the nozzle orifice diameter should be slightly smaller than the entrance diameter to the sprue. Nozzle orifice diameters larger than the sprue diameter will not release properly during part ejection and will leave a "cold slug" of plastic in the nozzle as well as creating an undesirable shear edge.

Proper sizing of the nozzle rear-openings is also important. The opening should precisely match both the diameter and taper of the discharge hole in the barrel end cap. If the nozzle opening is too small, a shear edge is created; and if the opening is too large, stagnation areas are formed. A properly size nozzle is shown in Figure 4.

Figure 3
Proper Nozzle Sizing



MATERIALS

The barrel end cap and nozzle should be constructed of hardened stainless steel. Both 17-4 PH. prehardened T440V or T420V have been used successfully for these applications.

Existing machine end caps are also commonly coated by the same process suggested earlier for protecting screws.

MOLDS

MATERIALS OF CONSTRUCTION

Stainless steel is the preferred material for mold cores and cavities for Fiberloc compounds. Although 420 type stainless steel or equivalent is generally recommended and most widely used, 414 prehardened stainless steel has been found to have good chemical resistance, is easily repairable, and can also be used.

The moving parts of the mold (knock outs, sliding cores, etc.) should be made from a hardened steel and plated. Metal galling can occur when two similar steels move in contact with each other. Prehardened and conventionally hardened steels are also acceptable mold materials, but require plating on all plastic contact surfaces.

Both hard chrome and electroless nickel plating have been used successfully in molds for Fiberloc composites. It is especially suitable for molds with deep or narrow mold recesses not accessible to chrome. Hard chrome plating has good resistance to corrosive and abrasive wear and is recommended over electroless nickel when possible. Unplated, hardened tool steels, aluminum and beryllium copper have all been successfully used for prototyping and limited production.

SHRINKAGE ALLOWANCES

Fiberloc polymer composites exhibit very low shrinkage, typically 0.000 in. to 0.001 per inch shrink. When strict part tolerances are required, the mold cavity should be oversized to allow for this shrinkage. The actual shrinkage encountered will depend on a number of factors including melt temperature, injection and holding pressures, gate size and wall thickness, and glass orientation.

DRAFT ANGLE

Required mold draft angle is dependent on part size and dimensions, but a minimum 0.5 degree draft angle is usually sufficient for untextured surfaces. If textured mold cavity surfaces are used, a standard minimum draft angle of 1 degree per side per each .001" depth of texture should be considered. Special textures will require greater draft angles, otherwise, the texturing will act as an undercut and can cause the part to stick in the mold.

VENTING

Localized burning and short shots can result when trapped air, gas or moisture vapor becomes super-heated under compression. Therefore, the mold should be adequately vented to allow for gas escape. Vents should be placed near weld lines as well as at the last areas of the cavity to be filled.

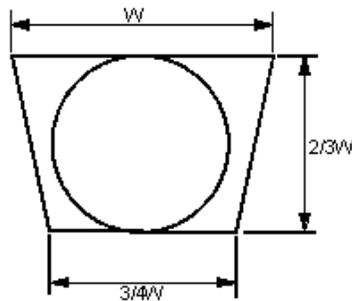
Typical vents are slots 0.6 to 1.3 cm (0.25 to 0.50 in.) wide, which lead from the mold cavity to the outside of the mold. They are typically (0.0015 to 0.0020 in.) deep for a short distance, (0.10 to 0.24 in.) near the mold cavity then increase in depth to a minimum 0.005 in. for the remainder of the distance to the outside of the mold. The vents are usually located on the mating surface of one of the mold halves. Venting may also be accomplished by grinding small flats on cores or knockout pins. In general, vents should be cut to a minimum depth initially then increased in depth if necessary.

SPRUE BUSHING

A sprue bushing with a standard 2-1/2° included angle, approximately 42 mm taper per meter (0.5 in. taper per foot), should be used. The entrance diameter of the bushing should always be slightly larger than the nozzle exit orifice. To promote a balance pressure to the runners and cavities, the exit diameter of sprue bushing should be larger than the diameter of the main runner.

The use of standard cast or wound heated sprue bushing is not generally recommended when molding Fiberloc compounds, but Kona heat tube technology hot sprues have been used successfully. Also, an excessively recessed sprue bushing is undesirable because it usually requires a long nozzle.

Figure 5
Relative Dimensions of a Trapezoidal Runner for Use in a Three-Plate Mold



RUNNERS

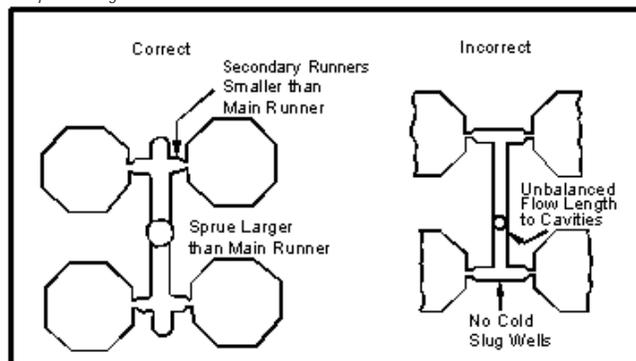
In a two-plate mold, full-round runners are preferred because they provide the highest volume-to-surface ratio, the least pressure drop, and are the easiest to eject from the mold. Depending on the part size and weight, typical full-round runner diameters are 0.6 to 1.0 cm (0.25 to 0.4 in.). Because of the excessive flow restriction, small diameter runners, less than 0.6 cm (0.25 in.) in diameter, should be avoided. Excessively large diameter runners offer little advantage and contribute to longer cycle times and greater material usage.

If a three-plate mold is being implemented, full-round runners are still preferred, but trapezoidal or half-round runners can be used. Rectangular runners are not recommended.

Figure 5 shows typical relative dimensions of a trapezoidal cross-section runner. The flow through a trapezoidal runner is equivalent to that of the largest circular runner whose cross-section can be inscribed within the trapezoid.

To maintain pressure and balanced flow during injection into a multiple cavity or multi-gated mold, the secondary runners should be slightly smaller in cross section than the main runner. Secondary runners should be perpendicular to the main runner, and the runner junction should be radlused and polished to remove burrs and sharp edges. Figure 6 shows a properly sized runner system.

Figure 6
Proper Running Sizer



COLD SLUG WELLS

During injection, the initial surge of material is generally cool since it has remained in the nozzle while the previous shot was being ejected from the mold. To prevent this cold material from entering the cavity and causing a visual defect, cold slug wells or runoffs should be incorporated into the runner system before material is allowed to enter the cavities. Properly sized runner systems designed for balanced flow which incorporate cold slug wells are shown in Figure 7.

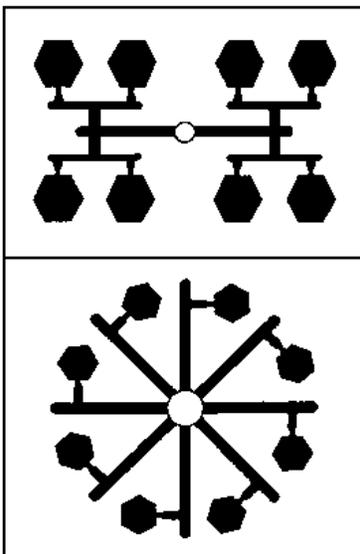


Figure 7
Runner Systems with Balanced-Flow
Cavity Layouts and Cold Slugs Wells

GATES

Fiberloc compounds have been molded satisfactorily through a wide variety of gate designs including fan, tab, edge and sprue gates. Submarine and pin gates have also been used but these latter two should be avoided except in small parts due to difficulty in gate break off, excessive gate wear, shear overheating and the possibility of glass breakage.

In general, glass reinforced materials should be run in molds with generous gates which approach the full thickness of the part and which have sufficient width

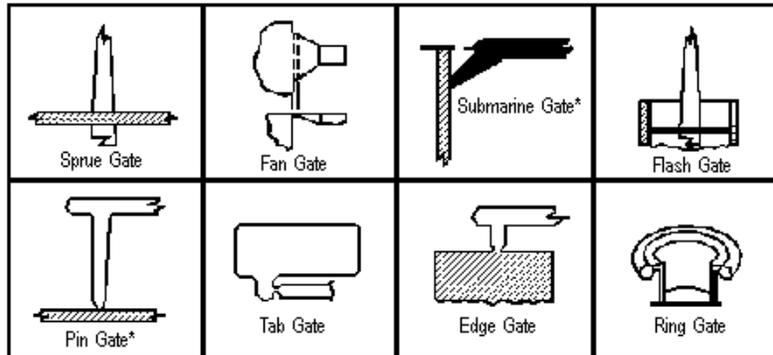
that the gate cross sectional area approaches the full cross sectional area of the runners. Small gates detract from part quality and can cause high stress and excessive shrinkage due to material overheating and premature freeze off. Smaller gates will also show more wear from abrasion. Molds designed with removable gates are a good idea for tooling designed for longer runs.

The land length of gate should be kept as short as possible. (0.03 to 0.04 in.) to reduce shear heating and promote the best combination of cycle times and injection speeds. Gates should be polished smooth with all rough edges and sharp corners removed.

Figure 8 illustrates several acceptable gate designs with round corners for minimum restriction.

In multi-gated cavities, the gate location and number of gates are very important to the appearance and performance of the molded part. Since gate areas are almost always more highly stressed due to orientation, gates should be located in non-critical sections of the part. Gating in thick sections of the part, allowing the material to flow to the thinner sections, keeps sink and splay marks to a minimum. If the design of the part requires a split in the flow front coming from the gate, a weld line will result where the flow fronts again meet.

Figure 8
Gate Designs



Consideration should be given to designing parts with a minimum number of gates. Multiple weld lines will weaken the part and could detract from surface appearance. Care must also be taken to design parts such that weld line will occur in non-critical areas of the part.

PROCESSING MATERIAL

Drying

Fiberloc compounds can normally be processed without predrying. However, during conditions of high humidity, moisture can condense on the pellets, causing problems such as excessive splay, foaming, low weld line strength, or other surface defects. If a moisture problem is suspected the compound can be dried using a hopper mounted dehumidifying air dryer. If a hopper mounted dryer is not available, then a tray oven can be used.

Drying conditions should be 82°C(160-180° F) for two hours. Material depth in each tray should not exceed 5 m (2 in.). Prolonged or overnight drying or higher temperatures are not recommended.

HEATER BAND SETTINGS

Heater band settings depend greatly on machine size, screw design and other settings, such as back pressure and screw RPM. Because of heat generated by shear, most molding machines yield melt temperatures higher than heater band settings.

For the initial trial of Fiberloc materials on machines over 450 ton clamp capacity, a starting barrel profile of 176°C(350°F) and a nozzle temperature of 193°C(380°F) can be used. A barrel profile of 182°C(360°F) and a nozzle temperature of 193°C(380°F) for machines less than 450 ton clamp capacity should be satisfactory. These settings should be adjusted to achieve an air-shot melt temperature which is 5 to 10°C(10 to 20°F) less than the desired temperature (more heat will be generated once the machine is cycling continuously). Since heat is being generated by the screw within the material, it is quite normal for the middle and front barrel temperature zones to slightly override the set point. Excessive override may possibly cause degradation of the material. As long as the machine is stabilized; i.e., cycling regularly, these set points do not require adjustment. Carefully monitor melt temperature with a needle pyrometer during initial start-up and after any condition changes.

Once heater band settings have been established and the desired melt temperature is achieved, subsequent runs on the same machine may be started at these conditions. Close monitoring of melt temperature is still recommended. Record processing conditions for future use.

SCREW RPM

For a screw of recommended geometry (see Screw Design section), a rotating speed of 40 to 50 RPM should be satisfactory. Due to increased diameter, a large screw has greater circumferential velocity than a smaller screw at a given RPM. The greater velocity promotes more shear heating of the molding compound. Therefore large machines generally require lower RPM at optimum conditions.

SCREW BACK PRESSURES

Screw back pressure will vary from machine to machine, but generally the back pressure should be in 0.35 to 0.70 MPa (50 to 100 psig) range. Low compression ratio screws may require higher back pressures to obtain proper mixing and melt homogeneity.

MELT TEMPERATURE

The melt temperature of the materials can be controlled by a proper combination of the heater band settings, screw back pressure and screw RPM. To develop ultimate physical properties, it is imperative that recommendations for melt temperature be followed when molding Fiberloc compounds.

The recommended melt temperature for processing these materials is 199 to 210°C (390 to 410°F). The melt temperature should never exceed 200°C (425°F).

To measure melt temperature at startup, use an accurately calibrated needle-probe hand pyrometer. When making a temperature measurement with a needle pyrometer, the molten material should be injected directly from the nozzle onto a piece of heavy cardboard or some other insulating material that will not readily conduct heat from the plastic. The injection pressure, injection speed and back pressure are usually adjusted to a lower setting for taking these air-shots than when at normal cycle; therefore, a melt temperature of approximately 5 to 10°C (10 to 20°F) lower than the recommended range is a good objective when starting. The needle should be jabbed into the molten plastic successively four or five times in different locations before the actual reading is taken. Remeasure melt temperature after processing conditions have stabilized and adjust process as needed to attain proper melt temperature.

Occasional wiping of the needle probe with some mold release agent will help prevent "freezing" of plastic on the probe during the initial portion of the reading. If material

"freezes" to the probe on the first insertion, it acts as an insulator on the probe's surface and erroneously low values for melt temperature will be obtained.

If gassing or bubbling of the hot plastic is observed during the air shot, it generally indicates that a higher than recommended melt temperature is being achieved, and can also indicate moisture in compound. Melt temperature should be rechecked. The plastic extrudate (rope) should appear smooth and reasonably glossy if the melt temperature is near optimum. In making this judgment of "glossiness" ignore the effect of the gloss on the surface appearance. Voids on the surface and in the core of the extrudate could also be indicative of a moisture problem.

INJECTION SPEED

A slow to moderate injection speed should be used at the start of molding run and increased to the point where the part fills and no signs of weld lines or sinks exist. If the injection speed is too fast, excessive frictional heat build up can result in velocity burning as the material flows through restrictions or over sharp edges. This frictional heat can result in surface appearance defects, brittleness or even degradation of the material. Injection speeds for air-shots should be relatively slow since there is very little resistance to the material flow.

INJECTION AND HOLDING PRESSURES

The amount of first stage injection pressure (booster pressure) that is required to fill the mold cavity will depend on the stock temperature, injection speed, mold temperature and mold design. Generally, pressures in the range of 50% to 70% of the maximum available offer the best consistency and process latitude.

It is advisable to start with lower pressures and increase to the desired pressure to avoid flashing the mold. The timer for the first stage injection pressure should be set to switch to holding pressure just as the part

is completely filled. This should coincide with the moment the screw completes its relatively fast forward travel leaving a 0.3 to 0.6cm (0.125 to 0.25 in.) cushion. Smear tip screws often continue to slowly creep forward and may eventually bottom out.

The second stage injection pressure (holding pressure) should be just enough to maintain a full part as the part cools and shrinks in the cavity. Holding pressure is typically 1/2 to 2/3 of the first stage injection pressure. Parts having thicker cross sections usually require greater holding pressure.

Overpacking the part with excessive holding pressure or time on the first stage injection pressure increases molded-in stress that is detrimental to properties. Generally, sink marks away from the gate indicate that more injection pressure/time is needed while sink marks near the gate indicate the more hold pressure/time is needed. Once it is apparent that gates are frozen off, hold pressure should be reduced to save on energy consumption.

A small cushion of material must be maintained ahead of the screw to compensate for part shrinkage as it cools under holding pressure, thus preventing sink marks. Ideally, the screw should only reach the full forward position after material movement has ceased.

MOLD TEMPERATURE

Accurate mold temperature control is essential for optimizing cycle times and finished part quality. Fiberloc compounds are usually run with mold heater-coolers between 20-65°C (70-150°F). Higher mold temperatures usually give improved surface appearance, better material flow, improved weld line integrity and lower part stress. Cooler mold temperatures give shorter cycle times. High pumping rates of the temperature control medium with minimal line and coupler restrictions will improve temperature control in the tool and optimize the combination of part quality and cycle time. Running the "B" half of the mold cooler than the "A" half usually facilitates part ejection and removal.

REGRIND

Runners and trim material can normally be

reused. Regrind materials should be kept very clean. Grinder blades should be kept sharp and clean to minimize heat buildup. To avoid thermal degradation, material should be cooled and stored below 150°F.

Regrind, at 10-30% levels, is commonly used in combination with virgin compound.

Degraded or contaminated regrind materials should not be used. It will only cause degradation of the good material used with it.

PROCESSING SUMMARY

In summary, to develop the ultimate physical and appearance properties of Fiberloc compounds, the material should be processed at the maximum allowable melt temperature without burning. It should be injected at a moderate speed, packed at the minimum pressure required to fill out the mold details and allowed to relax during the cooling stage.

START UP PROCEDURE

Prior to molding:

- Clean both mold halves thoroughly using a good recommended cleaner.
- Make sure all vents are thoroughly cleaned and free of any preapplied rust preventative, dirt or other material build-up.
- Address all water leaks and check for adequate water flow through the tooling.
- Clean and polish sprue bushing. Check for rough spots.
- Check nozzle and sprue orifice for proper match up.
- Review your process set up sheet for proper mold temperature settings. If none exists refer to this manual or call Geon CIM Technical Service Group.
- Set barrel temperature controllers to desired or recommended temperatures.
- Reduce injection and back pressures and screw RPM to the lower end of their operating ranges.
- Injection and back pressures should be checked and set during purging or after start of molding Cycle.

IMPORTANT DETAIL:

Purge the barrel with either GPABS, acrylic, styrene or an approved purge compound prior to introducing vinyl to the process. Use of polyethylene or polypropylene is not recommended.

NOTE: SEE WARNING CONCERNING ACETYL POLYMERS (next page)

- After barrel temperature settings have stabilized introduce the rigid Geon CIM Fiberloc to the machine.
- Take air shot melt temperatures and check using a hand held pyrometer and needle problem. If melt temperature is in the range of 200-220°C (390-410°F) proceed. If not, adjust heater band settings, screw RPM or back pressure to reach proper temperature. Recheck melt temperature after machine stabilizes in a production mode and maintain 200-220°C (390-410°F) melt.
- Observe molten vinyl appearance during the purging state. A smooth glossy surface is indicative of a good homogenous melt temperature. A smoking or frothy melt suggests melt temperature may be too high. Another good indicator of good melt temperature is the ability to draw down the hot rope into a thin monofilament. A brittle break indicates a low melt temperature. Porous or steaming may indicate moisture.
- Lightly, spray mold release into the core, cavity and sprue bushing, and commence molding vinyl into the mold.
- Start molding parts in the semiautomatic operation mode.
- Check and adjust injection pressures. Use medium range to start.
- Check and adjust injection velocities. Slow to moderate to start. Adjust up or down as needed.
- Adjust pressures and times to make acceptable parts.
- Adjust screw RPM and back pressure to obtain optimum melt temperature.
- Check heater zones for override and correct settings.
- Mold temperatures should be checked with a hand pyrometer and surface probe.
- Consult Trouble shooting Guide in this manual to correct defects in the molded part.
- If a sprue should hang up in the sprue bushing, never try to shoot through the hung up sprue to remove it. This may cause extensive shear heating leading to degradation of the vinyl.
- Intermittent purging of the vinyl is needed during machine stoppage over 15 minutes.

- See purging and shutdown procedures for proper mold maintenance after vinyl molding.

RECORD YOUR PROCESSING CONDITIONS FOR FUTURE USE

PURGING AND SHUTDOWN PROCEDURE

Rigid Geon Vinyls are susceptible to thermal degradation upon prolonged exposure to a high heat environment. Therefore, if there is an interruption in the molding cycle, the injection unit should be pulled back from the mold and the rigid Geon Vinyl should be processed through the barrel by making occasional "air-shots". If the delay is lengthy, the vinyl should be completely purged from the barrel with general purpose ABS or acrylic regrind. **AT THE END OF A MOLDING RUN, THE INJECTION MOLDING MACHINE SHOULD NOT BE SHUT DOWN WITH RIGID VINYL IN THE BARREL; IT MUST BE PURGED FROM THE BARREL WITH AN EFFECTIVE PURGE MATERIAL.** The carriage (barrel) of an injection molding machine should never be left in the forward position with the machine idle.

If rigid vinyl is accidentally overheated in the barrel, both the screw and barrel may have to be cleaned. If the condition is not severe, this may be accomplished by purging the barrel with acrylic or ABS at very low temperature. The "cold" acrylic or ABS will scour the barrel and remove any degraded vinyl material. If this method does not work, remove the screw from the barrel and clean mechanically.

In the event that a power failure occurs during the molding operation and the vinyl cools and solidifies in the barrel, special start-up procedures should be used when power is restored. Initially, the heater bands should be turned on low heat, 93 to 121°C (200 to 250°F) and held there until the material in the barrel has had time to warm through. The heater band setting should then be increased to slightly below normal operating temperatures. As soon as the machine reaches the higher temperatures, the screw should be manually jogged; and as soon as the material moves, the temperatures should be raised to those for normal molding conditions. The remaining material is then purged from the barrel with one of the

other recommended thermoplastic materials. At this time the rigid Geon Vinyl being used can be reintroduced into the barrel and production resumed.

Polyethylene or polypropylene are immiscible with vinyl and should not be used as a purge material. Flame retardant materials should not be used since they are susceptible to degradation themselves.

Care should be taken to minimize the possibility of rusting and corrosion when putting a mold into storage. The mold to be stored should be thoroughly coated with a commercially available spray to neutralize any residual acids present, dried and then sprayed with a commercial rust inhibitor, lubricant or mold release agent. At the end of a work day or over a weekend, it is also advisable to neutralize spray followed by a commercially-available mold protective solution of the mold core and cavity.

This temporary protective layer will be automatically removed by the first production shots.

WARNING!

It is important that vinyl compounds never come in contact with acetal or acetal copolymers (such as Delrin or Celcon*) within an extruder or molding machine, since at processing conditions the two materials are mutually destructive and involve rapid degradation of the products. Care must be taken to avoid even trace quantities of these materials coming in contact with the machine, as well as preventing any cross contamination of feedstocks.

If possible, vinyl compounds and acetal polymers and copolymers should never be processed in the same machine; however, when it is not possible, the machine must be purged with acrylic, ABS or other purge compound, followed by a thorough mechanical cleaning of the machine. For information on these materials, see the respective Material Safety Data Sheets.

*Delrin is a product of E.I. duPont de Nemours & Company, Plastic Product and Resin Department, Celcon is a product of Celanese Plastics and Specialties Company.

TROUBLESHOOTING GUIDE RECOMMENDED PROCEDURES FOR PROCESS UPSETS

Cycle Interruption - able to purge barrel.

- Restart press.
- Retract injection unit from the mold.
- Purge vinyl compound from barrel with vinyl compound.
- Check melt temperature before restart.
- Recheck barrel and nozzle temperatures.
- Resume normal operation.

Cycle Interruption - unable to purge barrel.

- Shut off heaters to barrel and nozzle.
- Resolve problem.
- Restart injection molding press.
- Set barrel and nozzle temperatures - 20°C (30°F) below normal operating temperatures.
- Start purging barrel as soon as reduced temperatures have been reached with vinyl compound.
- If contaminants are present, remove vinyl. Use purging compound until clean.
- Resume normal operation.

Slight degradation (Color shift and odor change) of vinyl compound in the injection molding press.

- Continue PVC molding operation.
- Recheck all nozzle and barrel temperature controllers.
- Recheck screw RPM and fill rate.
- Resolve if barrel temperatures are set too high, or if the controllers are malfunctioning.
- Resume normal operations.

Severe degradation (Charred compound and HCl odor) of vinyl compound in the injection molding press.

- Protect eyes, nose, and throat from hot release from degraded material.
- Retract Injection unit from the mold.
- Dissipate air around injection molding machine.
- Remove PVC compound from hopper.
- Put purge compound such as general purpose styrenics or acrylic in hopper.
- Purge barrel as rapidly as possible of degraded material into small portions (less than 1.5 pounds).
- Place degraded air shots in a bucket of water, if possible, and remove from the building.
- Locate errant barrel or nozzle temperature controller and repair.

- Determine other causes for degradation if machine is functioning properly.
- Purge barrel with small amount of vinyl compound to make sure controllers are operating properly.
- Resume normal operations.

TROUBLESHOOTING

Defect: Short Shots

Possible Causes:

1. Shot size too small
2. Injection pressure too low
3. Injection speed too slow
4. Cylinder temperatures too low
5. Mold temperatures too low
6. Insufficient venting
7. Sprues, runners or gates too small
8. Improper gate location
9. Melt temperature too low
10. Insufficient back pressure

Defect: Sink Marks or Excessive Shrinkage

Possible Causes:

1. Injection hold pressure too low
2. Shot size too small
3. Injection pressure too low
4. Hold time too short
5. Cooling time too short
6. Melt temperature too high
7. Mold temperature too high
8. Sprues, runner or gates too small (improper mold design)

Defect: Weak Welds

Possible Causes:

1. Mold temperature too low
2. Injection speed too slow
3. Melt temperature too low
4. Injection pressure too low
5. Insufficient mold venting
6. Improper gate locations and/or size
7. Cylinder temperatures too low
8. Screw back pressure too low
9. Nozzle diameter too small

Defect: Part Sticking in Cavity

Possible Causes:

1. Injection pressure too high
2. Hold pressure too high
3. Hold time too long
4. Core side of mold too hot
5. Rough surface on sprue bushing
6. Rough surface on cavity side of mold
7. Poor mold design
8. Undercuts

Defect: Blush Marks Around Gates

Possible Causes:

1. Mold temperature too cold
2. Injection fill speed too fast
3. Melt temperature too high or too low
4. Improper gate location
5. Sprue and nozzle diameter too small
6. Nozzle temperature too low
7. Insufficient cold slug well
8. Imperfections in gate openings
9. Moisture in the compound

Defect: Dullness on Molding surface

Possible Causes:

1. Barrel Temperatures too low (increase in small increments)
2. Screw back pressure too low
3. Injection fill speed too slow
4. Mold temperature too cold
5. Melt temperature too low
6. Moisture in the compound.

Defect: Silver Streaks on Part Surface

Possible Causes:

1. Melt temperature too high
2. Nozzle temperature too high
3. Injection speed too fast
4. Excessive moisture on material

Defect: Flashing

Possible Causes:

1. Injection pressure too high
2. Insufficient clamping pressure
3. Injection speed too fast
4. Melt temperature too high
5. Mold faces not plane and parallel
6. Improper venting (one cavity venting while another fails to fill)
7. Improper mold design

Defect: Dull Streaks, Flow Lines

Possible Causes:

1. Melt temperature too low
2. Runners too small
3. Improper gate size and/or location
4. Mold temperature too low
5. Inadequate cold slug wells

Defect: Warpage

Possible Causes:

1. Mold temperature too high (for thick wall sections)
2. Melt temperature too high
3. Insufficient hold time
4. Injection and holding pressure too high or too low

5. Injection speed too fast
6. Cycle time too short
7. Gates too small

Defect: Lamination

Possible Causes:

1. Purging compound left in cylinder
2. Mold temperature too low
3. Melt temperature too low
4. Injection speed too fast
5. Gate size too small
6. Injection pressure too high

Defect: Temperatures Over-Riding on Front Zones

Possible Causes:

1. Compression ratio of screw too high
2. Excessive back pressure
3. Insufficient air circulation on over-riding zones
4. Screw RPM too high

Defect: Burn streaks in Center of Sprue

Possible Causes:

1. Injection speed too fast
2. Front zone temperature too high
3. Screw RPM too high
4. Excessive back pressure
5. Compression ration of screw too high
6. Melt temperature too high

Defect: Burn Streaks at Gate

Possible Causes:

1. Injection speed too fast
2. Injection pressure too high
3. Gates or nozzle diameter too small (improper design)
4. Shear burning due to cold material

Defect: Discoloration or Burned Areas In Part

Possible Causes:

1. Injection speed too fast
2. Screw RPM too fast
3. Back pressure too high
4. Barrel temperatures too high
5. Faulty temperature controllers
6. Gates too small
7. Dead material hung up on screw or nozzle
8. Insufficient mold venting
9. Melt temperature too high
10. Moisture in compound

Defect: Weld Burning

Possible Causes:

1. Injection speed too fast
2. Insufficient mold venting
3. Melt temperature too high or too low
4. Screw RPM too high
5. Back pressure too high
6. Nozzle diameter too small
7. Sprue, runner or gates too small
8. Injection pressure too high
9. Excessive moisture on the material

PolyOne © Corporation, 1996

PolyOne © Corporation



<p>The information contained herein is believed to be reliable, but no representations, guarantees or warranties of any kind are made as to its accuracy, suitability for particular applications or the results to be obtained therefrom. The information is based on laboratory work with small-scale equipment and does not necessarily indicate end product performance.</p>	<p>and equipment used commercially in processing these materials, no warranties or guarantees are made as to suitability of the products for the application disclosed. Full-scale testing and end product performance are the responsibility of the user. PolyOne shall not be liable for and the customer assumes all risk and liability of any use or handling of any</p>	<p>control. POLYONE MAKES NO WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. Nothing contained herein is to be considered as permission, recommendation, nor as an inducement to practice any patented invention without permission of the patent owner.</p>
<p>Because of variations in methods, conditions</p>	<p>material beyond PolyOne's direct</p>	