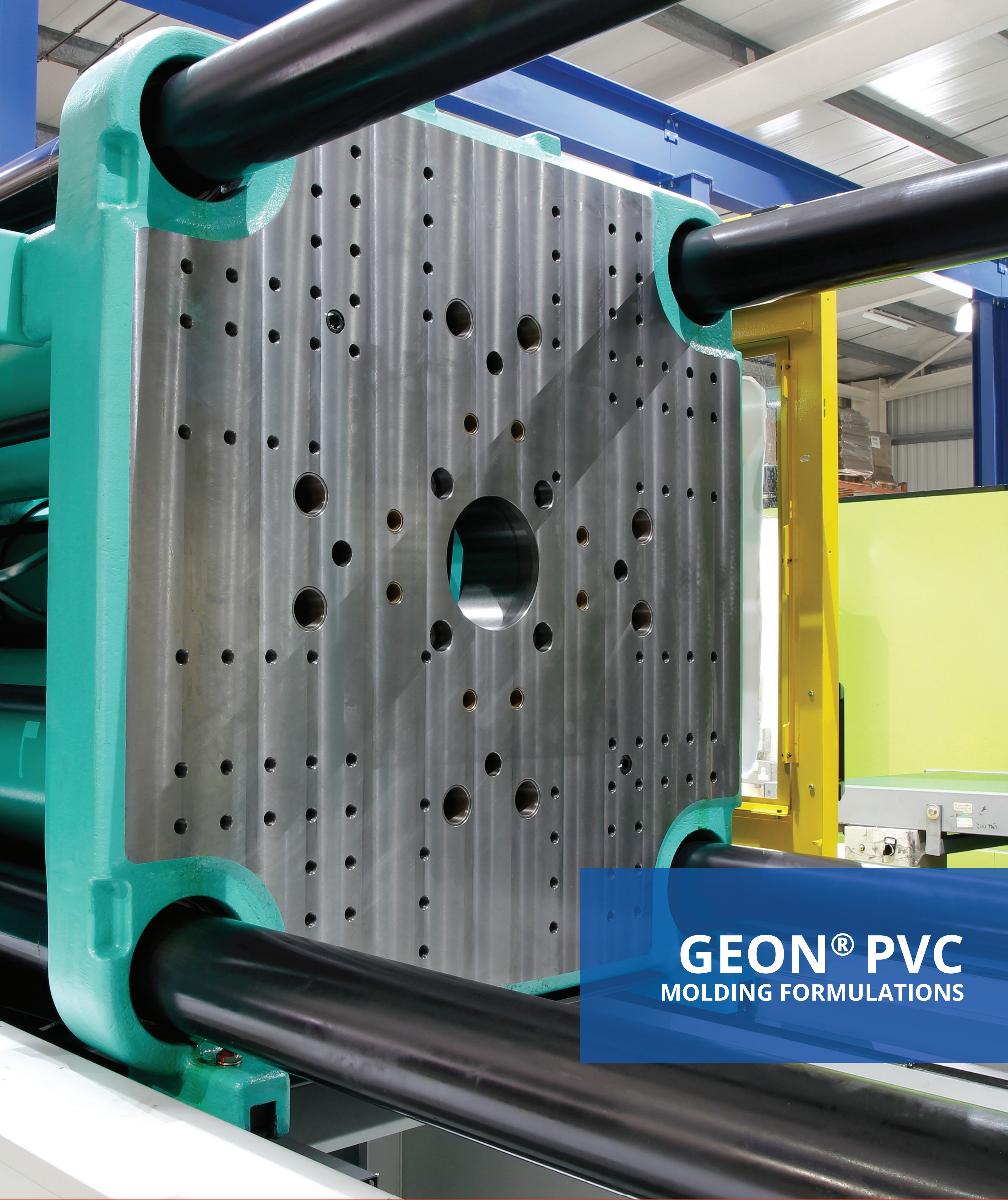


GEON® PVC

MOLDING FORMULATIONS

INJECTION MOLDING GUIDE

CONTENTS

INTRODUCTION

Polyvinyl chloride (PVC) is the third most produced thermoplastic in the world behind polypropylene and polyethylene, and hundreds of millions of pounds of rigid PVC are successfully injection molded around the globe each year. Because PVC is so versatile from a formulation and processing perspective, it can be injection molded into products for a variety of markets and applications.

Key Considerations for Successful Injection Molding

Melt Properties—PVC formulations are heat and shear sensitive at processing temperatures. This means the polymer melt can degrade if it is left at processing temperatures for extended periods of time or if it experiences high shear when flowing through restricted gates, runners, or part walls. The melt delivery system, which includes the molding machine injection unit and the mold runner and gating system, must be streamlined in design to keep material flowing and avoid stagnant areas or dead spots which can lead to degradation. The melt delivery system should also be designed to avoid highly restricted flow channels that can generate high shear. Injection molders who understand and follow these guidelines can successfully mold rigid PVC into complex shapes in a variety of sizes.

Molding Machine Design and Processing Conditions—Successful production of injection molded parts starts by converting pellets into a homogeneous, easy flowing melt. This guide provides recommendations for the appropriate design of molding machine components and processing conditions to produce an optimal melt for molding high quality rigid PVC parts.

Mold Design—Every polymer has different melt properties. In order to provide the greatest probability for success, molds should be designed with a specific polymer in mind. Factors such as metallurgy, gate and runner sizing, wall thickness, and part shrinkage can all change with polymer type. Contact GEON® and ask for a GEON Performance Materials Technical Service Representative to obtain more information about designing molds for use with GEON Vinyl Rigid Molding formulations.

Material Formulation—While many rigid PVC suppliers have standard flowing compounds for simple, thick walled shapes like pipe fittings, not all suppliers have high flow formulations for thin walled complex shaped parts. GEON Vinyl Rigid Molding formulations use state-of-the-art technology to provide enhanced flow and processing stability to fill a greater variety of larger, thinner walled, and more complex shapes. Contact GEON and ask for a GEON Performance Materials Technical Service Representative to determine the optimal rigid vinyl formulation
for your part design.

For more information, contact GEON:

Phone (Toll Free U.S.): 1-800-GET-GEON
Website: GEON.COM

As with all injection molded thermoplastics, care must be taken to follow recommendations for proper equipment and processing conditions to keep workers safe. This section includes health and safety information regarding injection molding rigid PVC.

WARNING: Acetal Not Compatible with Flame Retardant Polymers Like PVC

It is extremely important that PVC formulations and acetal or acetal copolymers never come in contact with each other at processing temperatures. At processing temperatures, the small amount of acidic vapors from PVC can catalyze the rapid depolymerization of acetal to form formaldehyde gas. The formaldehyde gas, in turn, causes further generation of hydrochloric acid. An accelerated chemical reaction continues and can ultimately lead to a violent release of gases.

Acetal and PVC need to be injection molded on different machines isolated from each other. If this is not possible, then thoroughly purge the molding machine between the two materials with general purpose ABS, acrylic, or other recommended purging compound and follow with a thorough mechanical cleaning of the barrel and screw.

Ventilation

It is necessary to have sufficient ventilation in any area where thermoplastics, including PVC, are injection molded. Sufficient ventilation includes an overhead roof exhaust fan and/or a sidewall exhaust fan. U.S. Occupational Safety & Health Administration (OSHA) requirements regarding "adequate" ventilation for molding operations must be followed in the United States and are recommended for other locations where no specific regulations exist.

Vapors During Processing

All thermoplastics emit vapors or offgasses when processed at melt temperatures. The identity and concentration of these offgasses around the injection molding machine depends on variables such as the material formulation, amount processed, processing conditions, and the effectiveness of the ventilation. Under normal processing conditions with OSHA approved ventilation, injection molding of PVC does not pose a health risk to workers.

Vinyl resin itself does not have an odor during normal processing. The slight odor emanating during injection molding of PVC comes from additives incorporated in the PVC formulation that enhance processing and physical properties.

PVC formulations are heat and shear sensitive at processing temperatures. This means the polymer melt can degrade if it is left at processing temperatures for extended periods of time or if it experiences high shear flowing through restricted gates, runners, or part walls. When rigid PVC overheats, it degrades, yellowing and generating gaseous hydrogen chloride (HCl). Chlorine gas and vinyl chloride monomer vapor are not generated when PVC degrades. HCl has a strong unpleasant acrid odor when present at low levels and irritates the nose and throat at higher levels. If you suspect material is degrading during molding, follow the procedures outlined in the **Processing** section under **Process Upsets**.

EQUIPMENT

Injection Molding Machine

Machine Type—Reciprocating screw injection molding machines designed to run engineering thermoplastics like ABS or PC are usually well suited for processing GEON® Vinyl Rigid Molding formulations. Avoid plunger machines or machines with an extruder combined with a plunger because they tend to have stagnant areas that lead to material degradation.

Machine Size—An injection molding machine having a minimum clamp force of 2 to 2.5 tons per square inch (0.0028 to 0.0035 MT/mm2) of projected area is typical. Thinner walls or long flow lengths may require higher clamp tonnage.

Barrel

Barrel Capacity—A shot weight which uses 50% to 80% of the barrel capacity is recommended for rigid PVC. A shot weight using 30% to 90% of the barrel capacity may be possible. However a long cycle time using a low shot weight percentage may lead to degradation of the material in the barrel. When a barrel's capacity is rated, it is rated by how many ounces of polystyrene the specific barrel will hold. Polystyrene has a specific gravity of 1.0 while rigid PVC typically has a specific gravity of 1.3 to 1.4. This means a barrel rated at 80 ounces of polystyrene will actually hold about 105 ounces of PVC.

Barrel Metallurgy—Ideally, the injection barrel is bimetallic, meaning there are two layers of different metals. The inner layer facing the melt is chemical and wear-resistant while the outer layer is a stronger steel sleeve. Single layer barrels that are only nitrided do not provide the best chemical resistance, but are commonly used when molding PVC and can work well if properly maintained. Barrel metals high in iron or cobalt are unacceptable for molding PVC because they cause pinking or streaking in the end product. Segmented or vented barrels are not acceptable for molding PVC because both types have stagnant areas impeding material flow (also called dead spots), which can lead to degradation and ultimately black specs in the finished part. An example of a very good recommended barrel would be a bimetallic barrel made from a nickel base alloy with high chromium content for good corrosion resistance against hydrochloric and other mild acid gasses. The barrel metal should also be compatible with nickel-based materials used to harden feed screw flights such as Colmonoy® 56.[1] Most screw and barrel manufacturers can supply recommendations for appropriate screw and barrel material combinations.

Screw

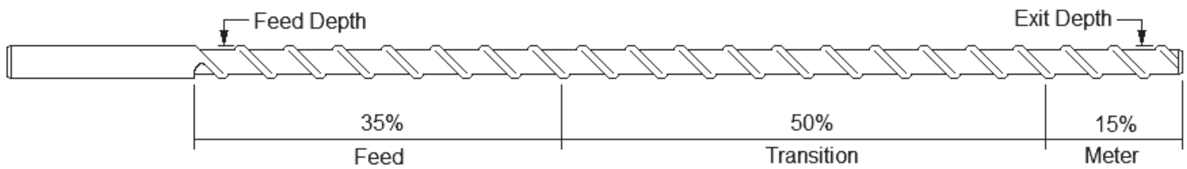
Screw Design—Compression ratio is defined as the depth of the first feed flight divided by the depth of the last metering flight. The recommended compression ratio for a rigid PVC molding screw is 2.2:1. While screws with compression ratios between 2.0:1 and 3.0:1 have been used successfully, it is recommended the compression ratio of the screw should be closer to 2.2:1.
As the compression ratio of the screw increases, the processing window for PVC may decrease. This is especially true when the compression ratio exceeds 2.8:1. A compression ratio less than 2.2:1 will tend not to squeeze all the air out of the melt, causing bubbles to be included in the shot.

---

[1] Colmonoy® is a registered trademark of Wall Colmonoy Corporation.

Screw geometry is a very important aspect to the overall performance of the screw. The recommended configuration for a rigid PVC molding screw is 35% feed, 50% transition and 15% meter.

Figure 1—Recommended Geometry for a Rigid PVC Screw



$$\text{Compression} = \frac{\text{Feed Depth}}{\text{Exit Depth}}$$

Table 1—Dimensions for Screw with a 2.2:1 Compression Ratio

Screw Diameter (in)	Feed Depth (in)	Exit Depth (in)	Screw Diameter (mm)	Feed Depth (mm)	Exit Depth (mm)
2.0	0.297	0.135	45	6.6	3.0
2.5	0.352	0.160	50	7.5	3.4
3.0	0.396	0.180	65	9.0	4.0
3.5	0.440	0.200	75	10.0	4.5
4.0	0.485	0.220	90	11.2	5.1
4.5	0.515	0.234	100	12.3	5.6
5.0	0.550	0.250	115	13.0	5.9
5.5	0.590	0.268	125	14.0	6.4
6.0	0.630	0.286	150	16.0	7.3
			160	17.4	7.9

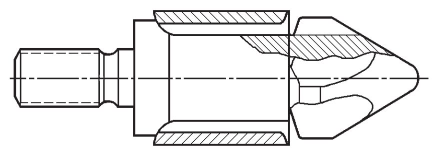
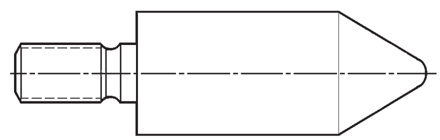
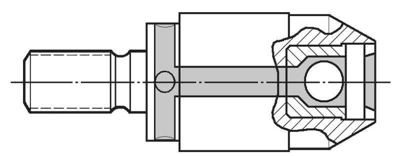
Screw Metallurgy—Stainless steel injection molding screws are highly recommended for use with rigid PVC formulations as they provide the appropriate combination of hardness and corrosion resistance. Screws manufactured with hardened tool steel (e.g. CPM® S90V) or screws manufactured with PH stainless steel for the root (e.g. 17-4 PH) and a nickel-tungsten alloy for the flights (e.g. Colmonoy® 56) are also acceptable.[2] If the screw is manufactured with tool steel such as 4140, corrosion protection such as triple chrome-plating is recommended. Irrespective of the metallurgy, all root surface areas need to be highly polished. Certain types of Stellite® metals are not recommended because they have a cobalt base which can cause streaking.[3]

Screw Tip

Screw Tip Design—The screw tip design is very important for rigid PVC formulations. Restrictive tips can cause shear burning. Like every component in the PVC injection molding system, it is important that the screw tip be as streamlined as possible to avoid stagnant areas where material could degrade. Free-flow sliding check rings with ample clearance under the check ring are strongly recommended. The clearance under the check ring should be 1 to 1.5 times the exit depth of the screw. This is the least restrictive screw tip that achieves complete shutoff during injection. Complete

[2] CPM® S90V is a registered trademark of Crucible Industries LLC and Colmonoy® 56 is a registered trademark of Wall Colmonoy.
[3] Stellite® is a registered trademark of Kennametal Stellite.

shutoff is important because it prevents backflow and delivers the full, steady injection pressure. Smear tips are not recommended for most GEON® high flow rigid PVC molding formulations as there is no mechanism for shutoff in this design, and the resulting backflow causes loss of injection pressure and increases residence time. Smear tips can be used with much higher viscosity PVC formulations designed for pipe fittings and other thick walled parts. Ball check tips are more restrictive than sliding check rings, and their use with rigid PVC is not recommended. They also tend to have stagnant areas where PVC can get stuck, causing a problem with black specks during processing.

Table 2—Screw Tip Types and Recommendations	
Sliding Check Ring ✓ Recommended for high flow rigid PVC formulations ✓ Recommended for PVC fittings formulations	 Sliding Check Ring
Smear Screw Tip × Not recommended for high flow rigid PVC formulations ✓ Recommended for PVC fittings formulations	 Smear Tip
Ball Check Screw Tip × Not recommended for high flow rigid PVC formulations × Not recommended for PVC fittings formulations	 Ball Check Tip

Screw Tip Metallurgy—The free-flow sliding check ring is one of the more important items requiring stainless steel construction. The screw tip constantly experiences shear so using stainless steel minimizes the need for resurfacing. While 420 grade stainless steel is ideal, other stainless steels such as 17-4 or CPM® S90V are also good choices.

End Cap

End Cap Design—It is important to prevent stagnant areas in the end cap. The flow channel should be tapered, radiused, and match the screw tip angle. The smaller the included angle of the end cap, the more streamlined the design is and the easier it will be to process any thermoplastic. However, streamlined end caps do require more maintenance to keep a high polish and prevent material from sticking to the end cap surface. An included angle of about 60° (30° each side) is a good compromise to the streamlined designs and requires less maintenance. End cap designs with long straight bores should be avoided because they create higher shear and restrict the material flow.

End Cap Metallurgy—The end cap should be made out of stainless steel, such as 420SS or 17-4. If it is not feasible to purchase a stainless steel end cap, triple chrome-plating should be used to protect the base steel.

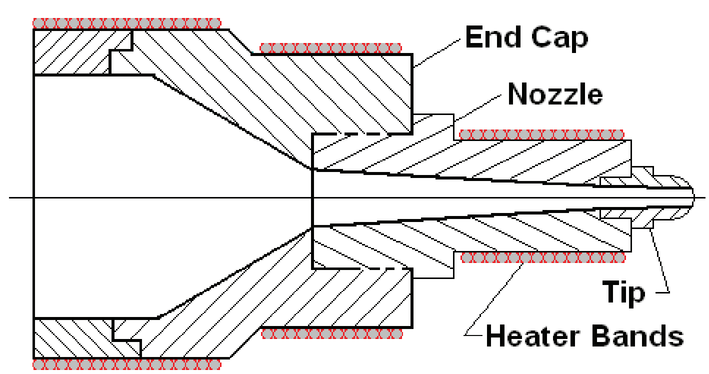
Nozzle

Nozzle Design—A nozzle length of 1 to 6 inches (2.5 to 15.2 cm) is suggested. Longer nozzle lengths may lead to shear burning. The minimum recommended exit diameter of the nozzle is 0.25 inch (0.64 cm)—for a 6 to 8 oz shot. As the shot size is increased, the nozzle exit diameter should also be increased. Nozzles with a full internal taper are preferred, although straight-bore nozzles are acceptable with shorter lengths. The nozzle tip diameter should be up to 0.04 in (1 mm) smaller than the rear opening diameter of the mold sprue bushing so that a complete seal is formed and no flash occurs.

Nozzle Metallurgy—Longer and more restrictive nozzles are more likely to result in shear burning. Because degradation accompanies shear burning, constructing the nozzles with stainless steel is an important preventative measure. For molders who do not injection mold PVC often or have short, non-restrictive nozzles, stainless steel may not be a feasible investment. In such cases, standard tool steel such as 4140 is acceptable. Nozzles made of 4140 must have their internal bores polished and well maintained.

Connecting Nozzle to Barrel—The barrel end cap should taper smoothly from the barrel diameter to the nozzle rear opening as shown in Figure 2. 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 triacthyristor circuit may be used. A variable transformer or an on/off relay control are not as effective for maintaining the processing control desired for rigid PVC.

Figure 2—Properly Connected End Cap, Nozzle, and Screw Tip



PROCESSING

Key Considerations

Developing Optimal Properties—To develop optimal physical properties and appearance of a rigid PVC formulation, the material should be processed at the maximum feasible melt temperature without degrading. It should be injected at a moderate speed, packed at the minimum pressure required to fill out the mold details, and allowed to relax sufficiently during the cooling stage.

Regrind—Runners, trim, short shots and other sources of clean PVC can be reground and mixed with virgin material. A regrind range of 10% to 25% is recommended, although higher regrind percentages up to 100% are being used in non-UL applications. It is important to maintain sharp grinding blades. Grinding generates significant heat so reground PVC should be cooled to below 150°F (66°C) before storing to prevent degradation of the material.

Melt Temperature—The ideal melt temperature range for GEON® Vinyl Rigid Molding formulations is 390°F to 405°F (199°C to 207°C). The best way to determine melt temperature is to pull the injection unit away from the mold and take an “air shot”. Measure the temperature of the melt with a calibrated needle probe pyrometer. In addition, the melt should look smooth and glossy. A dull melt indicates too low a melt temperature and a rough surface indicates either a high melt temperature or moisture in the melt.

Neutralize the Mold—Using neutralizer spray on the mold cavity surfaces is inexpensive and the best defense against tool corrosion. For best overall tool life, neutralize the mold cavity surfaces once per 24-hour day. Any tool, regardless of metallurgy, must be neutralized at least once a week. To properly apply neutralizer, spray all mold surfaces and runner blocks, with particular attention to the sprue bushing and other hard to reach spots.

The neutralizer spray must have a low moisture content to effectively work with PVC. It is also important that the neutralizer be strong enough to effectively neutralize hydrochloric acid residue left after molding PVC. Neutralizer also acts to protect the mold from corrosion caused by fingerprint acids and other damaging chemicals from the injection molding environment. Table 3 lists recommended neutralizer sprays and their suggested use.

Table 3—Recommended Neutralizer Sprays		
Spray	**Supplier Information**	**Comments**
PPE®[4] Step One Cleaner Acid Neutralizer and Dehydrator	Plastic Process Equipment, Inc. 8303 Corporate Park Drive Macedonia, OH 44056 www.ppe.com Toll Free: +1.800.321.0562 Phone: +1.216.367.7000 Email: sales@ppe.com	Recommended as a neutralizer during production and as the first step neutralizer for mold storage or short term shutdown
PPE® Step Two Rust Preventative	Plastic Process Equipment, Inc. 8303 Corporate Park Drive Macedonia, OH 44056 www.ppe.com Toll Free: +1.800.321.0562 Phone: +1.216.367.7000 Email: sales@ppe.com	Recommended as the second step rust preventative for mold storage or short term shutdown
Slide®[5] Acid Vapor Neutralizer Rust Preventative and Inhibitor	Slide Products, Inc. PO Box 156 430 S Wheeling Road Wheeling, IL 60090 slideproducts.com Toll Free: +1.800.323.6433 Phone: +1.847.541.7220 Email: info@slideproducts.com	Recommended as a one-step neutralizer/rust preventative for mold storage or short term shutdown

Recommended Purge Materials—The selection of a purge material depends on the effectiveness of the material in the changeover or cleanout and the cost of the material. Some purges may have a lower cost per pound but will require significantly more material to produce a clean changeover or cleanout. Table 4 shows a list of purge materials found to be effective with rigid PVC formulations for cleanout during production or for shutdown after production.

[4] PPE® is a registered trademark of Plastic Process Equipment, Inc.

[5] Slide® is a registered trademark of Slide Products, Inc.

Table 4—Recommended Purge Materials		
Purge	**Supplier Information**	**Comments**
Dyna-Purge®[6]	Shuman Plastics, Inc. 35 Neoga Street Depew, NY 14043 www.dynapurge.com Toll Free: +1.866.607.8743 Phone: +1.716.685.2121 Email: info@dynapurge.com	**Recommended for machine shutdown**—Dyna-Purge D2 works well for cold runner systems, especially to purge out the injection unit for shutdown. The machine is started back up with Dyna-Purge, which remains in the barrel during the down time. **Recommended for hot runners and sprues**—Dyna-Purge D2 works well with hot runner and hot sprue bushing molds. It can be run through the hot runner system or hot sprue bushing to clean out rigid PVC during production cleanout or for machine shutdown.
ABS, Polystyrene, or Acrylic (regrind)	Various	**Recommended for cleanout during production or for machine shutdown**—If it is not feasible to use a commercial purge material, virgin ABS and polystyrene or acrylic regrind can be effective purge materials for rigid PVC during production cleanout or machine shutdown. Use only general purpose natural grades which are free from colorants and flame retardant additives, ideally with a recommended melt temperature similar to that of PVC.

Hot Runner Systems

Hot runner systems have been used successfully in many applications with GEON® Vinyl Rigid Molding formulations. Having the proper hot runner system design and the appropriate vinyl formulation are critical to successful molding. Listed below are key points to keep in mind to be successful with running rigid PVC in a hot runner system.

- ✓ Hot runners must be used cautiously with PVC.
- ✓ Streamline runner channels in the manifold and drops to eliminate all stagnation areas.
- ✓ Minimize changes in flow direction to avoid shear heating.
- ✓ The volume of melt in the runners should be less than the volume of the part.
- ✓ The hot runner tip or drop should be a free flow type with external heaters.
- ✓ Hot runner drops should feed into a small cold sprue or runner before reaching the part.
- ✓ Use a hand held pyrometer and needle probe to verify the hot runner manifold and drop are controlling temperature properly.

[6] Dyna-Purge® is a registered trademark of Shuman Plastics, Inc.

- ✓ PVC must be purged from the hot runner system in the event of intermittent delays or process shutdowns.
- ✓ Purging with Dyna-Purge D2 works best for PVC hot runner systems.
- ✓ The injection molding press should have an ability to purge through the hot runner system with the mold in the open position.

Prior to Molding

Prepare the Mold

- ✓ Clean both mold halves thoroughly using a good recommended cleaner.
- ✓ Make sure all vents are thoroughly cleaned and free of any pre-applied 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 and size.
- ✓ Review your process set up sheet for proper mold temperature settings. If none exists, refer to this manual or contact GEON® and ask for GEON Performance Materials Technical Service.

Prepare the Molding Machine

- ✓ Set barrel temperature controllers to desired or recommended temperatures.
- ✓ Reduce injection pressures, back pressures, and screw RPM to the lower end of their operating ranges.

Prepare the Material

Drying—Drying is not usually necessary for rigid PVC. However, surface moisture can form on the pellet, especially in hot, humid summer months or in tropical locations. To reduce the need for drying, store material covered in a cool dry location. When drying is necessary, it should be done at 120°F–150°F (49°C–66°C) for approximately two hours.

Starting Equipment Settings

Heater Band Temperatures—Table 5 represents typical starting heater ban settings for molding rigid PVC. On most molding machines designed for engineering thermoplastics, these starting parameters will result in a melt temperature close to the recommended range of 390°F to 405°F (199°C to 207°C). The melt temperature should be measured by taking an air shot and measuring the temperature of the melt with a needle probe pyrometer. In addition, the melt should look smooth and glossy. A dull looking melt indicates a low melt temperature while a rough surface melt indicates a melt that is too hot and/or contains moisture.

The heater band settings in Table 5 represent an ascending temperature profile which is most commonly used. Sometimes a reverse or descending temperature profile is useful. It melts the pellets soon after they are fed to the screw, resulting in reduced abrasion, more efficient pumping of the melt forward, and reduced pinking or black streaks.

Table 5—Starting Heater Band Settings					
Machine Size	**Screw Diameter**	**Rear Zone (feed)**	**Middle Zone (compression)**	**Front Zone (metering)**	**Nozzle**
75 to 150 ton (68 to 136 MT)	1–2 in 25–50 mm	370°F (188°C)	370°F (188°C)	370°F (188°C)	370°F (188°C)
175 to 350 ton (159 to 318 MT)	2–3 in 50 – 75 mm	340°F (171°C)	350°F (177°C)	350°F (177°C)	360°F (182°C)
375 to 500 ton (340 to 454 MT)	3–4 in 75–100 mm	340°F (171°C)	345°F (174°C)	345°F (174°C)	360°F (182°C)
550 to 1,000 ton (499 to 907 MT)	4–6 in 100–150 mm	340°F (171°C)	345°F (174°C)	345°F (174°C)	350°F (177°C)
1,100 to 2,500 ton (998 to 2268 MT)	>6 in >150 mm	320°F (160°C)	320°F (160°C)	320°F (160°C)	350°F (177°C)

Screw RPMs and Back Pressure—Screw RPMs and back pressure work together to put work into the melt and greatly affect melt temperature. Table 6 shows the recommended starting screw RPM settings for a range of machine sizes. Screw RPMs and back pressure settings will depend on the screw compression ratio and screw configuration. More severe screw designs will require lower screw RPM and back pressure settings. Back pressure settings should start between 75 and 150 psi (5.2 and 10.3 Bar). Sufficient back pressure on the screw is needed to squeeze air bubbles out of the melt and provide a more consistent melt temperature, both of which lead to good shot-to-shot consistency.

Table 6—Starting Screw RPM Settings		
Machine Size	**Screw Diameter**	**RPMs Recommended**
75 to 150 ton (68 to 136 MT)	1–2 in 25–50 mm	50 to 75
175 to 350 ton (159 to 318 MT)	2–3 in 50–75 mm	50 to 75
375 to 500 ton (340 to 454 MT)	3–4 in 75–100 mm	30 to 50
550 to 1,000 ton (499 to 907 MT)	4–6 in 100–150 mm	20 to 30
1,100 to 2,500 ton (998 to 2268 MT)	>6 in >150 mm	10 to 15

Injection Velocity—The injection speed is dependent on the nozzle and sprue bushing diameters as well as the gate size and wall thickness. Moderate speed settings of 0.75 to 1.0 inch/sec (1.9 to 2.5 cm/sec) are reasonable to start. If shear burning is present in the part, reduce the injection speed. If no shear burning is present in the part and the part is not completely filled, increase the injection speed until shear burning is observed and then reduce injection speed in small increments until no shear burning is present. On molding machines with variable injection speed, it is sometimes helpful to use lower injection speeds until the material has filled the runner, gates, and initial regions of the part.

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 melt 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 processing 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 0.125 to 0.25 inch (0.32 to 0.64 cm) cushion. 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 ½ to ⅔ of the first stage injection pressure. Parts having thicker cross sections usually require greater holding pressure. Over-packing the part with excessive holding pressure or time on the second stage injection pressure increases molded-in stress that can be detrimental to physical properties. Generally, sink marks away from the gate indicate that more injection pressure or time is needed while sink marks near the gate indicate that more hold pressure or 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.

Mold Temperature—Accurate mold temperature control is essential for optimizing cycle time and finished part quality. Rigid PVC formulations are usually run with mold heater-coolers between 40-120°F (4°C-49°C). 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 easier part ejection and removal.

Start Up Procedure

1. IMPORTANT DETAIL: Purge the barrel with natural general purpose ABS, styrene, acrylic (regrind), or approved purge material (see Recommended Purge Materials) prior to introducing PVC to the injection unit. Use of polyethylene or polypropylene is not recommended because they are immiscible with PVC and will result in delamination of finished parts.
2. Injection and back pressures should be checked and set during purging or after start of molding cycle.
3. After barrel temperature settings have stabilized, introduce the PVC into the machine.
4. Take air shot melt temperatures and check using a hand held pyrometer and needle probe. If melt temperature is in the range of 380°F to 395°F (193°C to 202°C), proceed. If not, adjust heater band settings, screw RPM or back pressure to reach proper melt temperature. Recheck melt temperature after machine stabilizes in a production mode and maintain 390°F to 405°F (199°C to 207°C) melt.
5. Observe molten PVC appearance during the air shots. A smooth glossy surface is indicative of a good homogenous melt temperature. A smoking or frothy melt suggests melt temperature may

be too high. Porous or steaming melt may indicate moisture.

6. Lightly, spray mold release into the core, cavity and sprue bushing, and commence molding PVC into the mold.
7. Start molding parts in the semi-automatic operation mode.
8. Check and adjust injection pressures. Use medium range to start.
9. Check and adjust injection velocities. Slow to moderate to start. Adjust up or down as needed.
10. Adjust pressures and times to make acceptable parts.
11. Adjust screw RPM and back pressure to obtain optimum melt temperature.
12. Check heater zones for override and correct settings.
13. Mold temperatures should be checked with a hand pyrometer and surface probe.
14. 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 PVC.

Process Upsets

Expected Cycle Interruptions—Rigid PVC is susceptible to thermal degradation upon prolonged exposure to processing temperatures without moving through the barrel, runners, etc. Therefore, if an interruption in the molding cycle is expected to last longer than 15 minutes, the injection unit should be pulled back from the mold and the rigid PVC should be processed through the barrel by making occasional air shots. If the delay is lengthy, the PVC should be completely purged from the barrel with natural general purpose ABS, styrene, acrylic (regrind), or recommended purge material (see Recommended Purge Materials).

Unexpected Cycle Interruptions—In the event a power failure occurs during the molding operation and the PVC cools and solidifies in the barrel, the following procedures should be used.

1. Shut off heaters to barrel and nozzle.
2. Cool barrel with fans if emergency power is available.
3. Restart injection molding press when power is restored.
4. Set barrel and nozzle heater band temperatures to 250°F (121°C) for one hour.
5. Increase barrel and nozzle heater band temperatures 25°F (14°C) every 30 minutes until reaching 350°F (177°C).
6. When the barrel heater band temperatures approach 350°F (177°C), start jogging the screw until full rotation starts and then purge barrel with purge material.

Once the barrel is fully purged, the rigid PVC being used can be reintroduced into the barrel and production resumed.

Degradation During Molding—If you experience slight degradation of PVC from the barrel during molding (color shift and odor change), use the following procedure to eliminate the degradation.

1. Continue PVC molding operation.

2. Recheck all nozzle and barrel temperatures controllers to make sure they are within recommended guidelines and operating properly.
3. Recheck screw RPM and injection speed to make sure they are within recommended guidelines and operating properly.
4. Resolve these or other issues that can lead to higher than recommended melt temperatures or long residence times.
5. Resume normal operations.

If you experience severe degradation of PVC from the barrel during molding (significantly discolored material and strong, pungent odor of HCl), use the following procedure to eliminate the degradation.

1. Protect eyes, nose, and throat from hot release of vapors from degraded material. Wear a National Institute for Occupational Safety and Health (NIOSH) approved air-purifying, full-facepiece respirator with a chin-style, front- or back-mounted canister providing protection against the compound of concern.
2. Retract the injection unit from the mold.
3. Evacuate the air around the injection molding machine.
4. Remove rigid PVC from hopper.
5. Put purge material such as general purpose ABS, PS, or acrylic into hopper.
6. Completely purge barrel of PVC as rapidly as possible into small portions of melt (less than 1.5 lb/0.7 kg).
7. Place degraded melt into bucket of water and eliminate employee exposure.
8. Resolve issues with errant heater zone settings, defective controllers, or other issue that can lead to higher than recommended melt temperatures or long residence times.
9. Purge barrel with a small amount of rigid PVC to make sure machine is operating properly.
10. Consult site Environmental Health & Safety function prior to resuming normal operations.

Shutdown Procedure

When the molding of PVC has been completed, the injection molding machine should not be shut down with rigid PVC in the barrel because it is susceptible to thermal degradation upon prolonged exposure to processing temperatures without moving through the barrel. The injection unit should be pulled back from the mold and the PVC must be purged from the barrel with natural general purpose ABS, styrene, acrylic (regrind), or recommended purge material (see Recommended Purge Materials). Polyethylene or polypropylene are immiscible with PVC and should not be used as a purge material. Flame retardant materials should not be used since they also are susceptible to degradation.

If rigid PVC 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 natural general purpose ABS, polystyrene, acrylic (regrind), or recommended purge material (see Recommended Purge Materials) at the current temperature. If this method does not work, remove the screw from the barrel and clean mechanically.

1. Maintain production settings for barrel and nozzle heater bands, screw RPM, and injection velocity.
2. Retract injection unit away from mold, leaving ample room for purge to exit nozzle.
3. Empty and thoroughly clean the hopper of any PVC.
4. Empty the barrel of PVC by bringing the screw completely forward and running the extruder in the forward position until barrel is empty. Use high back pressure to maintain forward position. Approximately 500 psi (34 Bar) of back pressure will be needed depending on equipment type.
5. Feed purge material into the barrel and run extruder until PVC is flushed from the barrel.
6. Lower back pressure and manually make air shots at 25% of injection capacity to ensure barrel is clean and free of PVC.
7. Empty the barrel of purge material by bringing the screw completely forward and running the extruder in the forward position until barrel is empty.
8. The molding machine can now be safely shut down or used to mold another polymer except acetal. (If acetal will processed in the same machine as PVC, the injection unit must be disassembled and completely cleaned mechanically).

TROUBLESHOOTING

Sink marks	Part is not completely filled
• Not enough material injected in to the cavity • Injection pressure too low • Packing and hold pressures are too low • Packing and hold times are too short • Cooling time is too short • Reduce ratio of rib to wall thickness in part design	• Not enough material injected in to the cavity • Injection pressure is not high enough to overcome resistance in thin areas • Injection speed is too slow • Melt temperature is too low • Air is trapped in the mold • Part wall is too thin and material solidifies before filling the mold
Poor knit lines	**Gate blush marks**
• Melt temperature is too low • Mold temperature is too low • Injection speed is too slow • Poor venting in mold • Packing and hold pressures are too low • Adjust gate locations on part • Nominal wall thickness too thin	• Melt temperature is too low • Mold temperature is too low • Injection speed is too fast • Moisture in material • Hot spot in mold • Poor venting in mold • Sprue and gate diameters are too small • Insufficient cold slug well • Nominal wall thickness too thin
Silver streaks or splay	**Delamination**
• Melt temperature is too high • Injection speed is too high • Contamination in melt • Poor venting in mold • Gate diameter is too small • Nozzle and barrel temperatures too high	• Melt temperature is too low • Mold temperature is too low • Injection speed is too fast • Contamination with purge material • Gate diameter is too small

If these suggestions do not resolve your problem, contact your GEON Technical Service Representative or contact GEON and ask for a GEON Performance Materials Technical Service Representative.

Phone (Outside U.S.): 1-800-GET-GEON
Website: www.GEON.com

1-800-GET-GEON
SALES@GEON.COM