Molding Equipment

Vi-Chem PVC's can be molded on standard injection molding machines.

Screw Design

To avoid excessive shear of the polymer, screw compression ratio should not exceed 3.0:1. For a more uniform polymer melt and mixing, screw L/D (length to diameter) ratio should be at least 20:1 with a compression ratio of at least 2.5:1.
The compression ratio should be high enough to compress the low bulk density unmelted plastic into the solid plastic without air pockets. Too low a compression ratio will tend to introduce bubbles into the melt. Softer, flexible materials, it is recommended that a higher compression ratio screw is used (3.0:1) in order to flux the material.

Use the equipment manufacturer as a resource for recommended base material, flight O.D. material, and root protection.

Clamping Force

The force required to hold the mold closed against injection pressure depends on;

- Injection pressure to fill the mold and pack out the part.
- Injection speed required for filling the part.
- The dimensions of the part and runner at the mold parting line.
- Mold details – lateral slide actions, core pull requirements, etc.
- Precision and tolerance requirements.

Most well built molds can be adequately clamped if the machine has up to 4.0 tons/sq. in based on projected area. Since injection pressures are seldom over 1,400 psi, fill rates are moderate. The clamping force for PVC is recommended to be clamped at 1.5 – 2.5 tons/sq. in. of the projected area.

Molding Conditions

Melt Temperature

The melt temperature should be taken directly from the molten polymer (using a needle pyrometer) and should be checked periodically during production.
Optimum melt temperatures using Vi-Chem PVC is between 345 – 375 degrees Fahrenheit. Certain mold conditions may require higher or lower temperatures.

Cylinder Temperature Profile

A rising cylinder temperature profile (lower rear temperature) is normally preferred. Occasionally, a decreasing cylinder temperature profile will be used to help improve melt homogeneity.

Nozzle Temperature

The nozzle temperature should be adjusted to prevent freeze-off or drool. For optimum performance, it should be controlled independently from the barrel zones. To prevent drooling in certain cases, the use of pressure relief (suck-back) is recommended. Too much pressure relief can result in air bubbles in the melt stream.

Mold Temperature

Mold temperature should be measured with a thermocouple directly on the cavity surface. Recommended mold temperatures of between 70 – 130 degrees F. are recommended. Mold temperatures have little effect on mechanical properties. The main effect is shrinkage. See the effects of processing conditions on shrinkage of injection molded Vi-Chem Vinyl's.

Injection Speeds

Injection speed will vary with part thickness and geometry. In general, thin parts should be filled faster than thick parts. Moderate speeds are recommended. Excessively high fill rates can cause jetting; turbulence or severe shear heating that may result in burning and surface defects.

Pack and Hold Pressures

These pressures should be maintained for the time necessary to avoid sink marks and for the gate to seal off. Typically, Vi-Chem PVC should be 2-3 seconds for each mm of part thickness. This should also be adjusted to achieve minimal shrinkage.

Screw RPM's and Backpressure

A slow to medium screw speed is usually adequate. Minimal backpressure of 50 – 125 psi is recommended. Higher backpressure and screw speed can be used to improve mixing of the polymer.

Cycle Time

Molding cycle time is dependent on part size, geometry, thickness, polymer melt, and mold cavity temperatures, etc. Cycle times of 30 – 60 seconds are typical.

Mold Design

The following paragraphs stress some of the important aspects that should be considered when designing molds for PVC.

Material of Construction

Special materials are recommended for PVC. Stainless steel or chrome treating is recommended for mold construction. Other corrosive-resistant coatings are available. Use your mold shop, as they can be valuable resource.

Mold Surface Finish

Textured, Matte, and Highly Polished cavity surface work well with Vi-Chem PVC. Remember that PVC can give many different gloss variations, the mold surface must still be maintained to give consistent gloss. Venting is also an important factor in retaining mold surface, this will help in reducing gas build-up on the tooling.

Sprue Bushing Design

An incorrect sprue bushing frequently causes unnecessary cycle delays and surface defects. The diameter of the sprue at the smaller end should be equal to the diameter of the runner it feeds. A standard bushing should have a taper of at least 2.5°.

Properly mated injection nozzle and sprue bushings facilitate ejection of the sprue. The diameter of the hole in the nozzle should be 0.02 – 0.04 in less than that of the sprue bushing. Since Vi-Chem PVC's are elastomeric, sprue pullers with a generous undercut are needed for sprue removal. (e.g. "Z" pullers, sucker pins, or offset undercut).

Remember there should not be any sharp corners. This will reduce excessive shearing of the material.

Runners

Runners should be streamlined to reduce turbulence. A full round or trapezoidal runner should be used whenever possible to minimize pressure drop and for ease of removal. Runner systems should have a balanced layout. Runner section depends on the runner length, and the size of the parts. To improve flow and to facilitate ejection, the surface of the runners should be smooth, not polished. Cold slug wells, just before entering the gates are a must. Cold slug wells should also be at every point in the runner where the polymer changes direction. Again, there should not be any sharp corners.

Hot Runner Systems

Runnerless molding can be used with Vi-Chem PVC. It is important to work closely with the supplier of the hot runner system in order to optimize the design for the particular needs of each application. Some important items to consider when selecting or specifying a hot runner system are shown below.

Temperature Control

Sufficient heating capacity and control must be provided to ensure that neither freezing nor overheating occurs. Uniform temperatures throughout the hot runner system are important. Temperature controls should be capable of maintaining a temperature difference of no more than 5-10° F throughout the system.

Balanced Flow

Each branch of the runner system must have the same flow length; size and pressure drop to each cavity or part section.

Resin Hold-Up

The hot runners must have smooth contours and no areas of resin hold-up which can cause degradation of the polymer.

Sizing

Hot runners should be sized to minimize pressure drop. A reasonable limit should be used to keep the pressure drop in the runner system to less than 25% of the total pressure to fill the parts. Shear rate should be uniform in each branch of the runner. Excessively high shear must be avoided. Hot runner systems do increase the residence time of the polymer at melt temperature. The total polymer residence time, including machine and runner system, should be 5 shots or less.

Gate Types

Conventional rectangular, round, or tunnel gates can be used with Vi-Chem PVC. Wider fan gates or flash gates are recommended for applications where flowlines and distortion at the gate are an issue.
When using Tunnel gates the edges should be kept sharp to help break the gate. If the gate is too large in diameter or the edges are radiused, the gate may tear or difficult to break off.
Gate dimensions are important. Gates to small will require high injection pressure and will result in degradation, splay, and gate blush. Oversized gates may lead to longer hold time to avoid flow back, sink marks, gate seal off, and degating problems.
To avoid sink marks and filling problems, the gate should be located in the thickest section of the part.

Venting

Venting provides a path for the escape of air and other gasses in the cavity as the polymer displaces them. Inadequate venting of the cavity can seriously reduce flow into any cavity. Poor venting may result in pitting or corrosion of the metal surfaces of the mold. This is the result of repeated brief exposure to very high temperatures from rapid compression of air and gasses.
Venting problems may be aggravated by high melt temperatures, long holdup time, or holdup areas in the molding machine, which will generate more than normal amounts of gasses.
The vent opening into the mold should be broad but thin. Vent openings can be 0.25in wide, 0.040in land length, 0.020in deep vent channel, and vent depth can be .001-.002 to minimize the danger of flash.

Vents are positioned at the point of final fill to prevent burning of the part. Sometimes air entrapment can not be predicted before mold sample trial, and vents must be added after the molds are made for production.

Purging

Low-density polystyrene, Vi-Chem 85 durometer purge, as well as other compounds and specialty purging compounds can be used for purging Vi-Chem PVC. It is recommended that the injection unit is purged when the machine is shut down. To prevent cross contamination, proper clean up is always recommended before and after molding. It is not recommended an Olefin based compound is used for purging. **Under no circumstances should PVC and Acetel become in contact with each other. They form deadly gasses when combined.**

It is not recommended to purge through hot runners with any purging compounds. Purging compounds can stick inside hot runners and thus hindering the melt stream.

Categories of Vinyl Compounds

Generally, vinyl compounds are classified in three hardness ranges as measured by their shore durometers. Flexible vinyl's are read on the Shore A scale, while the semi-rigid and rigid vinyl's are read on the Shore D scale.

Flexible range	Shore A 40-90
Semi-rigid	Shore D 65-75
Rigid	Shore D above 75

The Effects of Processing Conditions on Shrinkage of Injection Molded Vi-Chem Vinyl's

Every vinyl injection-molding set-up has different and sometimes very special idiosyncrasies affected by the size of the shot, runner systems, and gating. The following are guidelines on controlling mold shrinkage:

1) The best way to control mold shrinkage is to design and manufacture the mold using the proper vinyl shrink factor. Use a shrink factor of 0.015-0.020 in/in. Be sure the mold shrinkage is calculated in the direction of flow. Lower durometers tend to shrink more than higher durometers.
2) Gating is of great importance. Generally the smaller the gate, the greater the shrinkage. A pin gate will give more shrinkage than a large fan gate.
3) Holding the part in the mold longer should reduce shrinkage and warp.
4) Lowering the mold temperature tends to reduce shrink, but may require increased melt temperatures and injection pressures to fill the mold.
5) Increasing the melt temperature should decrease shrinkage (less stress). Excessive heating can cause other problems.
6) Decreasing injection pressure and speed should also help reduce shrink.
7) Increasing filler in the compound may slightly reduce shrinkage, but specification, service, and color requirements limit this method.

It is important to realize that these are merely guidelines and each job has different demands and requirements. Nothing can replace experience, common sense, and the ability to experiment with different combinations.

NOTE: These are only guidelines. Under no circumstances will Vi-Chem be held accountable for part dimensions or tooling.