



Injection Molding Startup Procedure for Granular Polyester Molding Compounds

Prior to setting a mold into a press, it is necessary to first determine that the mold will fit in between the tie bars of the intended press. Once this is determined and before installation begins, the minimum clamp tonnage for the mold must be calculated. A couple reasons for the need to determine proper clamp tonnage are:

- Insufficient clamping force may lead to parts having unacceptable dimensions such as being too thick because the press may not have sufficient clamp force to force the material throughout the cavity(s)
- Potential mold damage from installing a mold that is too small for a press. Example: A mold that requires only 75T of clamp force is installed into a 400T press with non-adjustable clamp force may be damaged from too high of clamp pressure.

To determine the correct tonnage, multiply the projected area of the part at the parting line by 4,000-6,000 psi (27.6 - 41.4 MPa.).

Example: A part having a 12" diameter requires a minimum clamp pressure of 226T (2T/in²)

This can be calculated from the following formula:

$$\text{Clamp tonnage required} = r^2 * \pi * T/\text{in}^2$$
$$6''^2 * 3.1416 * 2(\text{T}/\text{in}^2) = 226 \text{ tons of clamp pressure}$$

In addition to reviewing the physical size of the mold, it should be determined if the shot size is within the press capacities. Typically, Plenco suggests using between 20% and 80% of the total shot capacity of the injection barrel. Typically under 20% utilization can lead to inconsistent processing while utilization over 80% can result in not enough shot capacity.

Once a mold has been matched with a press and is installed in that press, a standard procedure should be followed to begin molding parts. Following a written procedure each time a mold is installed makes it easier for the press operators by helping to minimize the omission of any procedural steps. After the mold is set the following startup procedure can be implemented.

1. Turn on the heat and frequently check the temperature of the molding surfaces with a calibrated pyrometer and surface probe. Typically, start with a mold temperature of 310°F - 350°F (154°C - 182°C). It is desirable to have as little temperature variance (typically within 10F) across the mold surface as possible.

2. Set the temperatures of the water jackets. Typically, the front zone should start at 180°F - 210°F (82°C - 99°C) and the rear zone should start at 150°F - 180°F (65°C - 82°C). Actual processing temperature settings may vary from the initial set points.
3. With the pyrometer and surface probe, check the “in” and “out” connections of each zone of the water heater to confirm that the actual water temperatures are close to the set temperature. There can be some variance from the set temperature, but a difference of 10°F - 20°F (5°C - 10°C) should be investigated. The problem may be calibration or something more serious such as a blocked water line. Of particular concern would be a situation where the “in” and “out” connections of the same zone are significantly different in temperature. This could be an indication that there is a blockage within the water jacket(s) and/or water lines. Corrosion can have similar effects as blockages.
4. The next step in the setup is to set the mold opening distance. This distance is important because if a mold opens too far, it can slow the overall cycle which will result in fewer shots per hour. The mold must open far enough to allow parts, sprue and runner system to fall freely into a box, conveyor or whatever device is catching them.
5. At the same time that the mold open distance is being established, the ejector stroke length should also be set. The ejection stroke should be just long enough to insure that the parts are ejected from the cavity. Additionally a multi ejection cycle aids in ejecting parts. The maximum ejector stroke should not hit against the ejector stop buttons.
6. Set the screw speed, which in most cases should be 60 rpm or less. A faster speed may not allow the material to be picked up and the result will be the same as running at a slower speed. In addition, running at a slower speed usually produces a more uniform stock temperature and a more consistent shot weight.
7. Set the back pressure for 50 psi (0.3 MPa) and air purge a couple shots. Check the stock temperature of the purged material from the third shot with a calibrated pyrometer and needle probe. The stock temperature is measured by air purging a shot of material and forming it into a ball, which is then checked after probing it 2 or 3 times using the needle probe of the pyrometer. The temperature should be 200°F - 210°F (93°C - 99°C). **NOTE:** At this point, the stock temperature will most likely be about 190°F - 200°F (88°C - 93°C) since you are just starting the process. It will take several molding cycles with a setback pressure before the stock temperature actually reaches 200°F - 210°F (93°C - 99°C). This is because actual backpressure is only realized when the nozzle is in contact with the sprue bushing.
8. PLEASE NOTE: **NEVER** breathe the mold when molding PLENCO granular polyester molding compounds. The reason for this is breathing will allow air to enter the mold which will stop the resin reaction before the material is completely cured. To prevent this from occurring, molds built for this type of molding compound are normally vacuum vented.

9. Before injecting the material for the first shot, the throttle or injection speed should be completely open. The amount of vacuum being pulled in the mold should be checked to insure it is at least 21" Hg and then set the amount of injection delay time needed to allow the vacuum system to achieve it. The shot size should be adjusted so there will be less than a complete shot. The injection pressures should also be set so the material will fill the cavities in 3 - 6 seconds. It normally is better to begin with a less than full shot (about $\frac{3}{4}$ to $\frac{7}{8}$ of a complete shot) and gradually work up to full shots. This way, there is less chance of damaging the mold by bending or breaking core pins and it will show if the filling of the cavities is balanced. Additionally you will not flash the tool so no vacuum ports will be plugged with flash.
10. Just prior to injecting the first shot, the mold should be waxed. Carnauba wax works well for this purpose. To wax a mold, melt the wax on the molding surface and with the aid of a small natural bristle paintbrush, spread it over the entire molding surface, getting it into every pocket and corner. Remove any excess wax from the mold surface, be sure to mold immediately or wax will burn on to the tool surface.
11. The molding parameters should be adjusted to produce good parts from all cavities, each shot. Typically, the injection time should be 3 - 8 seconds. The primary injection pressure should be in the range of 900 - 1,600 psi (6.2 - 11.0 MPa) while the secondary injection pressure should be set at $\frac{1}{2}$ to $\frac{2}{3}$ of the primary injection pressure. After an acceptable molding process is established, it should be capable of continuing without change for many hours.
12. To insure that typical batch to batch variations of the material will not affect the processing setup, 200 psi (1.4 MPa) is added to the primary injection pressure. The throttle or injection speed is adjusted so it takes the same length of time to inject the material. Example: With the throttle fully opened, the injection time was taking 7 seconds. With the injection pressure increased by 200 psi (1.4 MPa), the throttle is adjusted so it still takes 7 seconds to fill the mold.
13. Because a reciprocating screw does not pick up exactly the same amount of material every cycle, it is a good practice to use a cushion when injection molding thermoset materials. To establish a cushion, $\frac{1}{8}$ " (3 mm) is added to the injection stroke and at the same time the switch to secondary pressure or holding pressure is set at $\frac{1}{4}$ - $\frac{3}{8}$ " (6 mm- 10 mm) from the end of the stroke. Machines that use timers are more difficult to control because the injection time will vary from shot to shot. As a result, the switch from primary pressure may not be timed correctly, which can adversely affect the parts.
14. Once the material has been injected into the mold, it is held under pressure until the gate is cured. The time it takes the gate to cure can vary and the larger the gate opening the longer it will take to cure. The most common indication of an under cured gate is depressions or sink marks at or near the gate or on the part.

15. After the gates have cured, the screw can be run to plasticize material for the next shot.

Typically, the screw should finish running one or two seconds before the press opens to eject the parts from the present shot. If the screw is returning too early, the screw run time should be delayed. Once the material is plasticized, the amount of time the next shot of material is held in the barrel should be kept to a minimum. The longer the plasticized material is held in the barrel, the more likely it is that the molder will have processing problems such as nozzle freeze ups, short shots, or heavily flashed parts.

16. Upon ejection from the mold, the sprue should have a soft bulbous tip. If it does not, too much heat is being transferred from the sprue bushing to the nozzle and the probability of nozzle freeze off increases significantly. In order to prevent this, we suggest doing one or more of the following:

- Install a device to gently blow a continuous stream of air over the tip of the nozzle.
- Install a nozzle that has a smaller radius than the sprue bushing, for point contact and less heat migration. An example is $\frac{1}{2}$ "R nozzle to $\frac{3}{4}$ "R sprue bushing.
- Insert a piece of **corrugated** cardboard between the nozzle and sprue bushing to insulate the nozzle.

Please note, in general, we do not encourage the use of a sprue break, as it does not pull the tip and can lead to other processing problems.

Date Printed: November 10, 2015

Date Revised: November 18, 2014

Supersedes Revision Dated: October 8, 2007

This information is suggested as a guide to those interested in processing Plenco Thermoset molding materials. The information presented is for your evaluation and may or may not be compatible for all mold designs, runner systems, press configurations, and material rheology. Please feel free to call Plenco with any questions about PLENCO molding materials or processing and a Technical Service Representative will assist you.