Optimizing PET Preform Manufacturing-

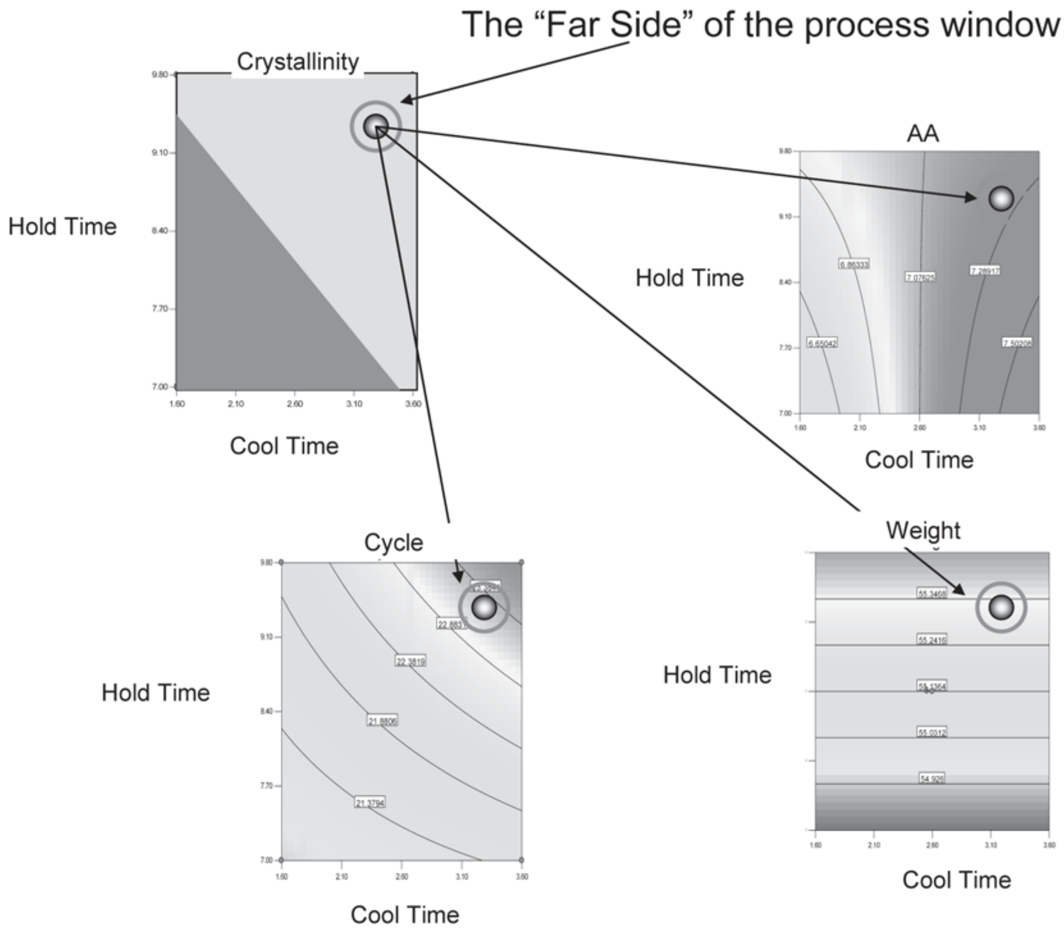
Guidelines for Finding the Starting-Point

Stephen Deneka

As the world's leading provider of Six Sigma training provider for the PET container manufacturing industry, we spend a lot of time working with our clients to find an optimum starting point for injection molding process optimization. Clients often ask us to rectify troublesome processing problems. Be it trying to process light-weighted preforms in blowmold, AA problems or high cycle times in injection, we have to come in and achieve results quickly. Our chosen method for doing that is statistical Design of Experiments (DOE). No other method allows someone to learn and improve a process as quickly and effectively as DOE.

For injection projects, we often find that the process is set in a far corner of the processing window. It often makes sense when the technicians explain how they got there; usually to compensate for a mold maintenance issue that was causing a defect. But sometimes after rectifying the mechanical problem, the plant continues to process from that far corner.

We then have two choices; we can execute a series of DOE's from the current process, which will get us where we need to go with confidence and evidence. Or we can jump to a more logical base processing point before starting the experiments. Due to time constraints and the base of experience from having executed hundreds of DOE's in the past, we often choose the latter. Now the tough question… what's a logical base processing point?



Limitations and Nuisance Factors

There are often some limitations that we can not correct in a short process assistance visit. Maybe it is a hot runner on its last legs, an undersized dryer or unusually low water flow. These limitations will affect what we can achieve with just machine adjustments. Compared to modern molds, older molds often have poorer venting and cooling than the latest and greatest from the world's better manufacturers.

Nuisance factors are issues that we can not control that can affect the end result. It could be a problem with one of the two dryer desiccant beds where a setting will work great for four hours one bed, but not the other. Or perhaps the company is always buying short supplies of off-spec resin at spot prices. Nuisance factors affect our ability to hold an optimized process.

Correcting and controlling limitations and nuisance factors widens our process window. So here we go with guidelines on how to get to a roughly set up PET preform process.

First things first… filling the cavity properly

The main factors involved in filling the cavity are the Shot Size, Injection Pressure (or Velocity on closed loop machines), the Transition Position, Hold Time and finally Hold Pressure.

1. Set the Shot Size to maintain a 5mm cushion of plastic at the end of the hold phase.
2. Set the Transition Position to 15% of the Shot Length (Shot Length = Shot Size – Cushion)
3. Set the Injection Pressure or Velocities to reach the Transition Position (Injection or Fill Time) in 10% of the part weight. (e.g.: A 45gram preform should start with a 4.5 second injection or "fill" time. A 28gram preform with 2.8 sec.)
4. Flatten the injection pressure curve as per diagram #1. The pressure (in blue) stays relatively constant as we reduce the velocity (in red) as we near the shot size. We don't want a pressure peak as indicated in diagram #2.
5. Use three steps in the Hold Pressures. Set the first hold pressure to 60% of the peak injection pressure, the second hold pressure to 50% and the third pressure to 40%. (e.g. If the peak injection pressure is 1500psi during injection, Hold Pressure #1 should be 900psi, #2 = 750psi, #3 = 600psi)
6. Set the Hold Times of the three phases to a total of 10% of the gram weight. Start by dividing the time equally for each stage of hold. (e.g.: a 45 gram preform should start with a total of 4.5 seconds of hold time; 1.5 seconds for each of the three stages.)

Injecting like the curve on the right, without lowering the injection pressure as the cavity fills, generally creates excess flash and makes it harder to pack out the cavity. Excessive injection velocity will exceed the ventilation capacity of the cavity and trap air against the sidewalls. You want to keep an even flow front to push the air out through the thread vents during the injection phase, not overpack it out in the hold phase, leading to higher likelihood of gate failures. If your hold time is almost twice your injection time in order to barely eliminate sinks, your injection velocity is likely too high.

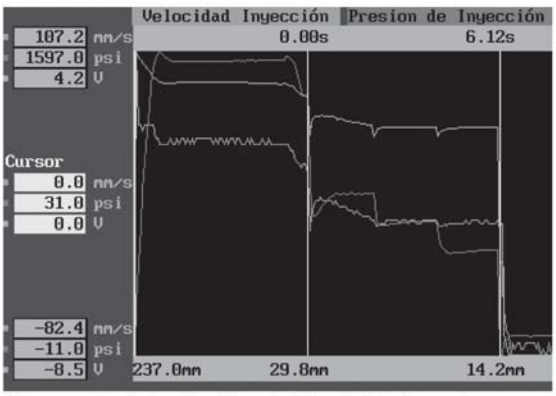


Diagram 1: Injection from left to right of graph, pressure shown in blue, velocity in red.

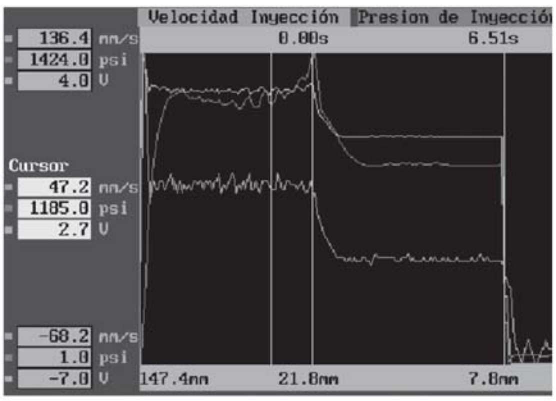


Diagram 1: Peaked pressure curve with flat hold curve.

Using 10% of the gram weight for injection time is a guideline. You'll find that with thin wall product (<2.5mm) you can usually inject faster. At >4.5 seconds thickness, you may have to go even slower.

Cooling Time

For a multitude of reasons, we want to minimize cooling time. Higher IV, lower AA, higher profit. Preforms can exit the robot with surface temperatures over 60^{o}C and not suffer deformation in a gaylord that isn't overfilled. So, we then lower the cooling time until we are on the edge of gate deformation, part transfer problems, or deformation on the exit belt.

Extrusion and Mold Heat

Now with the (usually) faster cycle, it's time to see if we really need all of that heat that we're putting into the process. The less heat energy we put into the resin, the less we have to take out later with cooling time. First goal is to reduce the Shot Size back time to 2 seconds or less. Sometimes we find machines with 5 seconds of idle time, meaning that the transfer pressure was far more than what the process needed. Excess transfer pressure often adds to over-rides on the middle extruders.

Next start dropping your back pressure 100psi at a time. If you have a drum style colorant mixer, make sure the blender motor is running even if you aren't dosing color to prevent idle pellets in the drum. Reducing back pressure is also key to reducing over-rides. Once you encounter bubbles, go back up 100psi.

Stephen Deneka, *an ASQ Certified Quality Engineer, has over seventeen years experience in PET processing. He is fluent in English and Spanish and has helped processors in the Americas, Europe and Asia improve their profitability and quality.*

Next start lowering your extruder temperatures and machine heats (barrel head through to the nozzle) 5^{o}C at a time until you encounter unmelts, then go back up about 7 deg C on the extruders. You may want to leave your distributor section hotter than the rest of the heaters in the area to facilitate valve movement.

Next, start lowering the mold manifold heats 5 deg C at a time. You'll know that you've encountered the limit when you see splay / streaks of plastic in the sidewalls. Go back up 5 deg C.

Finally, drop the tip heat no more than 5% of the original value at a time. (e.g.: Tips normally at 40%? Lower them 2% at a time.) Watch for gate crystallinity (including in the cross-section of the gates), long gates, and excessive gate peeling.

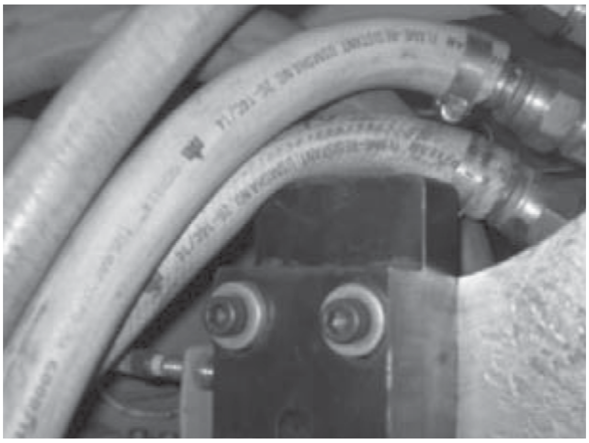
If you've removed a significant amount of heat from the process, you can then try to shave more cooling time. You might be surprised by how many tenths you can save in cooling time by running your extruders, manifolds and tips a little bit cooler.

Lubrication and Clamp Movement

A commonly overlooked area is clamp end auto-lubrication cycles. We often find lubrication intervals set to the maximum that the machinery manufacturer permits. By lowering lubrication intervals to the 200-250 cycle range, we can often improve the smoothness and speed of the clamp at its slow-down points. 0.3 seconds is possible. Less grease, more frequently, gives more control than a large amount every 1250 cycles.

"Stealing time" from the ejector dwell time is a common method for reducing cycle that can be avoided completely by purchasing a photo-eye system for the mold when it is brand new. Photo-eyes can allow the machine to bring the ejector back at least 0.1 seconds sooner. On a machine with a 20 second cycle at 95% efficiency, that's an extra 7259 cycles per year. On a 10 second cycle, it's 14518. If it's a 144 cavity mold, that's an extra 2,090,592 preforms per year, and in addition, there will be far fewer stoppages for part transfer problems.

Mind your hoses! A common problem with three axis robots are the hoses that get crimped and eventually cut with constant movement and fatigue. Vacuum leaks require extra ejector forward delay.



The Next Steps

Your process should be in a better place now and ready to start proper optimization with Design of Experiments. Our preference is for a series of two-level, half-fraction, five factor experiments with four center points. A good series of experiments is similar to the optimization order that we explored here:

***#1:* Refine the injection:** Injection Velocity, Transition Position, Hold Pressure, Hold Time, Cooling Time

#2: Refine the melt: Extruder temperature, Back Pressure, Mold Temperature, Tip %,

#3: (Optionally as a Response Surface Methodology experiment) Improve the gate appearance: Valve Gate Close Delay, Final Hold Pressure, Cooling Time, Tip %, Pullback.

#4: Optimize drying: Process Temperature, Drying Time, Airflow. Be sure to measure resin moisture, AA and IV as responses to best learn about your drying process. (You can usually partner with your resin supplier on the IV testing. Moisture testing in this regard is best conducted with a true moisture analyzing method rather than the gravimetric loss method.) ☯