Cycletime Tips - General

Volume 38: A Way to Optimize an Injection Molding Process - Part One

By Mark Shade

Introduction

This paper describes a structured procedure for identifying process parameter windows for the plastic conditions of injection molded components. The generally accepted four plastic conditions are Plastic Temperature, Plastic Viscosity (Flow Rate), Plastic Pressure and Plastic Cooling Rate.

Our objective is to find the process center for the four plastic conditions then center the part characteristics at the process center. Think of this as driving up a narrow mountain road lined with sharp rocks on one side and steep drop off on the other. Most of us will seek to stay as far away from the danger as possible, the center. Too often the plastic processor is asked to hug one edge or the other of the process 'lane' and is constantly battling the rocks or trying to crawl up from over the edge. This procedure will help you find that confidence zone, the center.

Some Review

Plastic Temperature is the actual measured melt temperature as it is injected into the mold. The most common method for measuring melt temperature is by using a pre-heated handheld pyrometer into a purged shot. Plastic temperature is affected by several process parameters such as machine tuning setpoints, cycle time, material-drying temperature, screw rotate speed, backpressure, and many others. It is not uncommon to find the actual melt temperature to be 50°F and occasionally 100°F different from barrel pyrometer setpoints. For this reason, you must measure actual melt temperature to insure that it is in the middle of the supplier recommended range.

Plastic Viscosity is the relative viscosity of the polymer as determined by a Melt Flow Curve (also called an On-Machine Rheology curve)1. The Melt Flow Curve will help us determine the optimum injection fill rate for the process. The plastic viscosity of the injected polymer will be affected by melt temperature, fill rate, nozzle orifice size, runner system and gate sizing.

Plastic Pressure as developed in the mold cavity will affect part size, part weight, shrinkage, some part aesthetics and to some extent, part cooling. An ideal molding process includes cavity pressure sensing ability to directly measure this value. However, this tool is not available to many processors and by generating a plastic pressure loss study across the mold, the processor can gather very good plastic pressure information to help identify where the biggest pressure losses are within the polymer flow path.

Plastic Cooling Rate defined as the rate at which the heat energy is removed from the molded plastic component. Thermoplastic injection molding is an energy transfer process. Energy is required to soften the plastic pellets sufficient to allow the polymer to flow. A little additional energy is added to the plastic as it encounters shear through the nozzle, runner system, gate and part. The plastic cooling rate is affected by mold design, coolant temperature, mold piping, packing pressure and significantly by coolant flow rate. If the melt temperature is changed, the cooling rate and cooling time needed will change in response to the additional heat energy imparted. Remember that a significant amount of this heat, possibly 40%, is still in the part when ejected from the mold so post mold handling is also critical to production of molded articles and must be considered as a continuation of plastic cooling rate.

The Procedure

Pick the mold temperature that will result in the desired part appearance and physical properties. Be advised that mold temperature and the resultant cooling rate can have an effect on the physical properties of some polymers like polyphenylene sulfide (PPS), polyetheretherketone (PEEK) and most semi-crystalline polymers. You should consult the material manufacturers' information to select your mold temperature strategy. For molds using water as the coolant, remember that turbulent coolant flow will produce the most efficient and fastest rate of heat exchange from the molded article away from the mold. Turbulent flow is defined as when the flow velocity is to the point where the fluid exhibits complete mixing of the coolant molecules. For water cooling systems, the approximate minimum flow in gallons per minute for turbulent flow in typical mold cooling channels is listed below.

NPT Pipe Size	Minimum Flow Rate/GPM
1/16	0.35
1/8	0.45
1/4	0.55
3/8	0.75
1/2	0.90
3/4	1.20

Review the mold coolant channels and plumb the available coolant to achieve the greatest heat removal from the hottest area(s) of the part(s). Note that more heat energy is removed from the areas of the part that shrinks onto cores and other features. Note also that thick areas of a part will require greater heat removal than thinner sections of a part.

By choosing the manufacturers' recommended mold temperature, insuring turbulent coolant flow and directing coolant at the hottest areas of the mold, the Plastic Cooling Rate is optimized. The cooling time will be optimized later in the process.

Achieving a target melt temperature that is mid-point of the manufacturers' recommended range would optimize the Plastic Temperature. Actual melt temperature will have to be checked periodically throughout the optimization process as adjustments are made to many process parameters that affect actual melt temperature. The forward zones of the barrel, Front & Nozzle as well as a heated sprue or a hot runner manifold should be set in a manner to only maintain target melt temperature.

Backpressure is usually set between 800 - 1200 psi plastic (80 -120 psi hydraulic on a 10:1 intensification ratio machine) to keep the screw flights full of plastic. For improved melting, use the slowest screw RPM that returns the screw 1 - 3 seconds before the cooling timer ends each cycle. If you experience poor colorant mixing, unmelted pellets or difficulty maintaining melt temperature using these guidelines; you may want to take a closer look at the screw design. Melt Uniformity screw designs offer improved color mixing at lower colorant loading, a more consistent and uniform melt with fewer hang-up areas within the screw flights. Excessive backpressure and high screw RPM can contribute to material degradation, elevated melt temperatures and color shift due to excessive heat, all factors contributing to an inefficient process.

At this point we have optimized two of the four plastic conditions. In part two of this CycleTime Tip, we will spend a significant amount of time to optimize what may be the shortest portion of the cycle; mold filling.

1 - A Procedure for Establishing Proper Fill Rate for Single or Multiple Cavities an On-Machine Rheology Curve