F I M M T E C H , I n c .

Frontier Injection Molding and Material Technologies, Inc.

www.fimmtech.com Suhas@fimmtech.com (760) 525 9053

ESTABLISHING A ROBUST INJECTION MOLDING PROCESS, PART 2: DETERMINING THE CAVITY BALANCE.

Importance of Cavity Balance:

As the plastic flows through the runners into the cavity, the melt has a given temperature, pressure and velocity. All these three variables are time dependent, which means that the value of each one of them will change within the short time the plastic flows till it reaches the end of fill. For example, melt temperature drops with time. If the melt temperature is T deg F at the start of injection, then after one second, the melt temperature is lower than T deg F. The final quality of the part being molded is a function of each of all these variables or in other words, the final dimension and quality of the part depends on the temperature, pressure and velocity of the plastic as it fills the cavity.

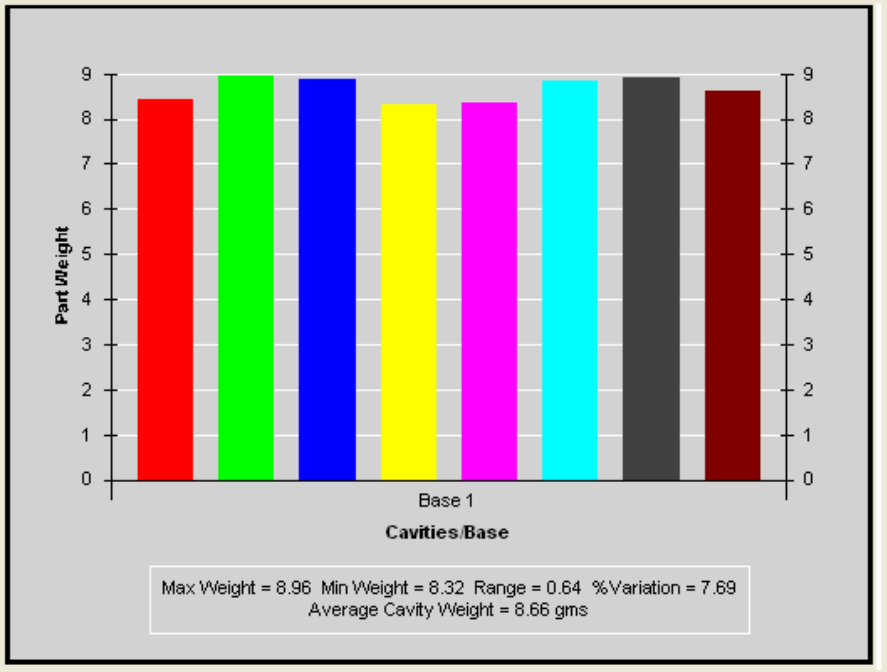
Consider a one cavity mold. The melt temperature at the end of fill is 450 deg F, the plastic pressure is 8000 psi and the velocity at which the plastic entered the cavity was 4.5 inches per minute. This produced a part with a certain dimension and finish. Now, if the temperature drops to 400 deg F, the part will shrink less and will produce a part that is now larger than the previous shot. Similarly, if the end of fill pressure and velocity changes, there will be a change in the dimension and/or finish of the part. Now consider a two cavity mold with identical parts being molded in each cavity. If the two cavities do not fill under similar conditions, then based on the above discussion, we know that the two parts produced from each cavity will be different. That is why, to produce identical parts, it is imperative that the flow in the two cavities is balanced, demonstrating the fact that the plastic has reached the end of the fill under the same conditions and will result in identical parts. This is the importance of cavity balance.

Procedure to determine the cavity balance:

1. Set the holding pressure to zero.
2. Set the holding time to zero.

3. Set the screw recovery delay time to about a value close to an estimated holding time.
4. Set the cooling time to a value such that you know that the part will be cool enough to eject.
5. Set your injection speed to the value obtained from the Viscosity Curve study.
6. With the rest of the settings the same as you had in the viscosity study, start molding.
7. Only by adjusting the transfer position, mold parts that are just short. If there is a visible cavity imbalance, then the 'biggest' part should be just short.
8. Make three such shots and take the average weight of each cavity and plot a graph.

A typical graph from an eight cavity mold is shown below.



How to Use this Information:

(The assumption here is that the gate and runner sizes are the same.)

Check the %variation between the maximum and the minimum fill cavities. In most cases, the % variation should not be greater than 5%. For tight tolerance parts, the variation should not be more than about 3%. If the tolerances are large, variation more than 5% is acceptable. More importantly, it is the final quality of the part that should be checked to see it there is need to tighten up the cavity balance. For example, the Cpk values of all the parts from all the cavities is a good measure. The following should be considered:

a. Amorphous materials can tolerate more imbalance than crystalline materials.
b. Tighter the tolerances, lesser should be the variation.

Venting is a very big contributor to plastic fill and can have a big impact on the cavity balance although the gate and runner sizes are the same. Make sure that the venting is the same for all cavities.

About FIMMTECH:

FIMMTECH is a consulting firm that provides services in the area of Injection Molding of Plastics. FIMMTECH is also in the process of developing products that will increase efficiency of the molding process, educate personnel and better manage the molding facility. One of the first products to be released is the software 'NAUTILUS' that helps in the development of robust and optimized processes. Suhas Kulkarni also teaches a course on Injection Molding at the University of California, San Diego that can be offered as In-House seminars. For more information please visit www.fimmtech.com

Thank You.
Sincerely
Suhas Kulkarni.