

**FIMMTECH, Inc.**

**Frontier Injection Molding and Material Technologies, Inc.**

**www.fimmtech.com**

**Suhas@fimmtech.com**

**(760) 525 9053**

---

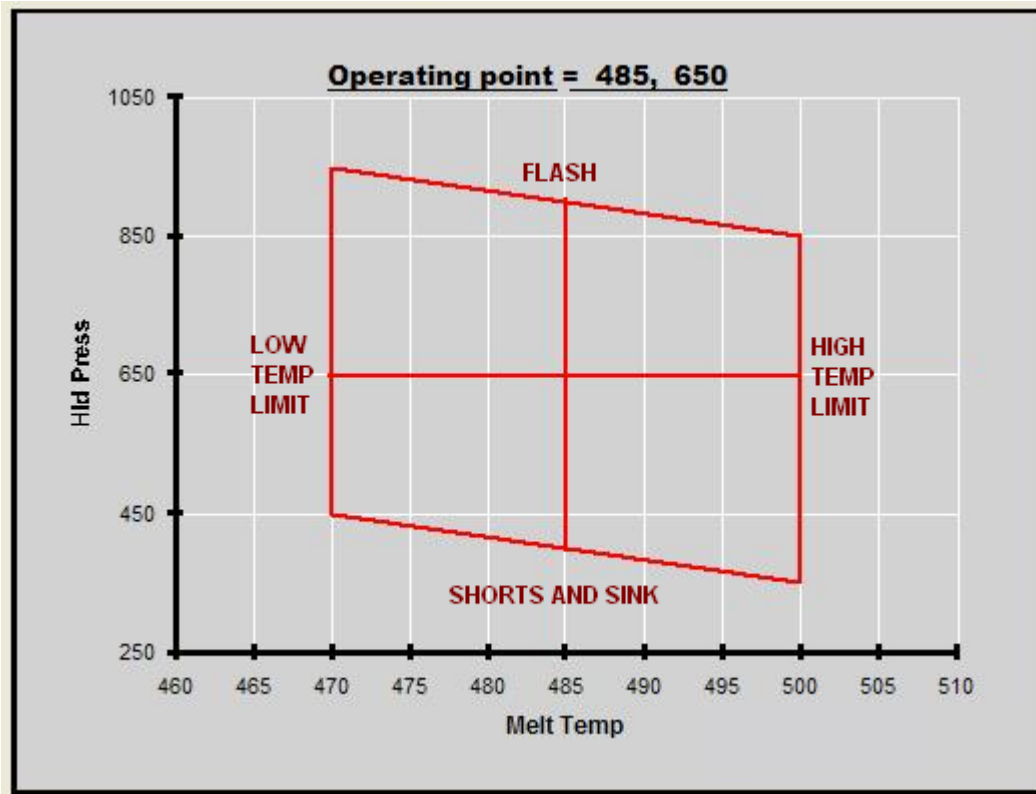
**ESTABLISHING A ROBUST INJECTION MOLDING PROCESS, PART 4:**  
**THE HOLDING PRESSURE PHASE.**

**Introduction to the Holding Phase:**

The injection of the plastic into the cavity can be divided into two main phases. The first phase is the injection phase. The mold cavity is completely filled with the molten plastic in the injection phase. The second phase is the holding phase. The holding phase follows the Injection phase. The holding pressure must pack into the cavity, the plastic equivalent to this volumetric shrinkage that occurs during the cooling down of the plastic as it hits the cold walls of the mold. There are various parameters that need to be controlled during this phase. These are packing pressure, holding pressure, packing time and holding time. In most cases packing and holding are not differentiated and are collectively called the holding phase. This type of molding where the injection phase is separated from the holding phase is called de-coupled molding, a term coined by RJG Technologies.

The packing phase consists of packing the cavity with plastic equal to the theoretical weight of the part. The theoretical weight is equal to the density of the plastic multiplied by the volume of the cavity. Any less plastic will result in an under-packed part and any more will result in an over-packed road. Under-packed parts results in defects such as sinks and voids in the parts. Such parts usually exhibit post molding shrinkage. Over-packed parts can have in-built stresses that usually get relieved after molding resulting in defects like warpage or premature failure. This section will deal with the pressures related to the holding phase. The next section will deal with the times related to the holding phase.

The ideal holding pressure is determined by evaluating the Process Window of the mold. The Process Window is also called the Molding Area Diagram. This is the area in which acceptable parts are molded. The bigger the window, the robust is your process. Refer to the picture below.



In the picture above, outside the process window, the parts can either have sink or flash above or below the holding pressure limits. Outside the melt temperature limits, you may have unmelted plastic below the lower limit or degraded plastic if you go over the limit. Both these are detrimental to the part quality. The process is set to the center of this window so that any variations within the window would still yield acceptable parts. The larger the window, the more robust is the process.

#### **Procedure for determining the Process Window.**

1. Set the barrel temperatures to attain the lower value of the recommended melt temperature.
2. Set the injection speed to the value obtained from the viscosity curve experiment.
3. Set all holding times and pressures to zero
4. Set the cooling time to a value more than what would be typically necessary. E.g. If the estimated cooling time is 10 seconds, set the cooling time to 20 seconds.
5. Start molding and adjust the transfer position to make a part 95 to 98% full.
6. Let the process and the melt stabilize by molding the parts for about 5 to 8 shots.

7. Now set the hold time to a value such that you are sure the gate is frozen. (In the next section we will learn how to optimize this time.) This would be based on some previous experience. For example, for a 30% glass filled PBT or Nylon with a gate size of 0.070” this time ends up between 6 and 8 seconds. In this case, for this part of the experiment, we would set the hold time to 10 to 12 seconds.
8. Increase the holding pressure in small increments and note the pressure where an acceptable part is made (no shots, flash, etc. ...).
9. Note down this pressure as the ‘Low Temperature - Low Pressure’ corner.
10. Increase the pressure further in similar increments and note down the pressure where there is an evidence of an unacceptable situation such as part sticking in the mold or flash, warpage, etc. .... Note down this pressure as the ‘Low Temperature - High Pressure’ corner.
11. Repeat steps (9) and (10), but at the high end of the recommended melt temperature. This time the two extreme pressures would be the ‘High Temperature - Low Pressure’ and ‘High Temperature - High Pressure’ corners.
12. Joining these four corners would now generate the process window or the molding area diagram.
13. Set the process to the center of this window.

Note: In case of crystalline materials, the recommended melt temperature range is small. In such cases you can perform the window study at one temperature only and generate a high and low limit for the holding pressure.

### **How To use this information:**

The process window is an indicator of how much can you vary your process and still make an acceptable part. An ideal situation is you have a wide process window. If the process window is very narrow, then there is always a danger of molding parts with defects. For example in the above graph, if the process window is very small, then one could get occasional short shots or occasional flash due to the natural variation in the process. A robust process is one that has a large process window and accommodates the natural variation.

### **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](http://www.fimmtech.com)

Thank You.

Sincerely

Suhas Kulkarni.