What's different about Universal Molding™?

Some of the Universal Molding™ differences are:

At Universal Molding™, we pride ourselves on our innovative approach to injection molding. Our commitment to excellence has led to several key differentiators that set us apart:

1. **Standardization**: During optimization, we ensure that every setup is nearly identical. No longer does a good setup belong to just one individual. We've developed systematic procedures to ensure consistent setup parameters across all molders. Standardization is our goal.
2. **Molding from the Desk**: Before even touching the equipment, mold, or material, we meticulously determine equipment sizing and initial parameters. This proactive approach streamlines the entire process.
3. **Machine Rheology**: Our evaluation of flow behavior goes beyond traditional viscosity and shear rate equations. By utilizing Injection Time and Peak Power equations, we accurately describe the dynamics, resulting in up to a 70% reduction in injection time determination compared to other methods.
4. **Molding with Graphics**: Our simplified method allows anyone, not just molders, to evaluate processes effectively. Quality control personnel find this tool invaluable.
5. **Accelerated Learning Curve**: Our intensive training program transforms novices into precision molders within just three months. The results have exceeded expectations.
6. **Parameter Effects**: We've dissected cause-and-effect relationships. For instance:
    - **Mass Dimensions**: These effects depend on material quantity and are manipulated during the Holding stage.
    - **Thermal Dimensions**: Shrinkage-related effects come into play during the Cooling stage.
7. **Molding Language**: We communicate clearly, describing the impact of each parameter with simplicity and precision.

What % should be filled during the injection stage?

At Universal Molding™, we have adopted the practice of filling approximately **95%** during the injection stage, including the weight of the runners. Let me explain the reasons behind this decision:

1. **Issues with 98% Filling**: In the past, we recommended filling between **95% and 98%**. However, we discovered that filling up to **98%** caused more problems. Beginners often thought that more was better and ended up facing issues such as flash, screw bounce, and limited pressure usage.
2. **Focus on Ideal Speed**: During the injection stage, we aim to fill around **95%** of the mold's required volume within a defined injection time. Although we mold parts rather than runners, we consider the mold filling difficulty within an ideal injection time. This time encompasses everything from the recovery position to transfer, including both **runners and parts**.
3. **Benefits for Molders**: We have consistently promoted the **95%** filling approach, including runner weight, for over **5 years**. The benefits have been significant for both novice and experienced molders.

Ultimately, each company can choose its approach based on experience and needs. Those who continue to fill between **95% and 98%** without considering runner weight can continue doing so. However, for those starting in this field or managing a large number of molds, we recommend adopting Universal Molding's approach for equal or better results.

How significant is a change in decompression?

A change in **back pressure** would influence **discharge density**, leading to variations in **% utilization**, **total volume % during the injection stage**, and the **cushion**.

Example:

1. **Initial Values** (Back pressure = 150 bar):
   - Recovery position = 51 mm
   - Transfer position = 9 mm
   - Shot weight = 15.8 g
   - Cushion = 8 mm
   - % Volume during injection:
     $$\frac{\text{Area (RP – T)}}{\text{Area (RP-C)}} \times 100\% = \frac{(51\,\text{mm} - 9\,\text{mm})}{(51\,\text{mm} - 8\,\text{mm})} \times 100\% = 97.7\%$$
   - % Utilization:
     $$\frac{\text{Area (RP – C)}}{\text{Area (RPmax)}} \times 100\% = \frac{(51\,\text{mm} - 8\,\text{mm})}{120\,\text{mm}} \times 100\% = 35.8\%$$
   - Total Volume:
     $$(2.5\,\text{cm})^2 \times \frac{3.1416}{4} \times (5.1\,\text{cm} - 0.8\,\text{cm}) = 21.11\,\text{cm}^3$$
   - Discharge density:
     $$\frac{\text{Mass}}{\text{Volume}} = \frac{15.8\,\text{g}}{21.11\,\text{cm}^3} = 0.75\,\text{g/cm}^3$$
2. Back pressure = 75 bar (same recovery and transfer positions):
   - Cushion = 7 mm
   - % Volume during injection:
     $$\frac{(51\,\text{mm} - 9\,\text{mm})}{(51\,\text{mm} - 7\,\text{mm})} \times 100\% = 95.4\%$$
   - % Utilization:
     $$\frac{(51\,\text{mm} - 7\,\text{mm})}{120\,\text{mm}} \times 100\% = 36.67\%$$
   - Total Volume:
     $$(2.5\,\text{cm})^2 \times \frac{3.1416}{4} \times (5.1\,\text{cm} - 0.7\,\text{cm}) = 21.60\,\text{cm}^3$$
   - Discharge density:
     $$\frac{15.8\,\text{g}}{21.60\,\text{cm}^3} = 0.73\,\text{g/cm}^3$$

Summary: By lowering the back pressure:

- Cushion decreased by 1 mm.
- % Volume during injection decreased by 2.3%.
- % Utilization increased by 1%.
- Discharge density decreased by 0.02 g/cm^3.

Comment: The part weight did not change due to back pressure because the mass was compensated by holding pressure (mass dimensions).

How significant is the discharge density?

The **melt density in injection molding** is a necessary value when determining screw positions; however, the value obtained from raw material suppliers can be off by over 20%. Thermoplastic melts are compressible, and their density is influenced by the **pressure** and **temperature** of the melt. During recovery, parameters such as **back pressure** and **barrel zone temperatures** affect melt density. Additionally, determining **recovery positions** becomes complicated if some melt passes to the other side of the **check ring** during injection. For this reason, **Universal Molders** prefer **Discharge Density**.

Discharge Density:
- Discharge density is more accurate when determining the recovery positions.
- This density depends on several factors:
    - **Mass**
    - **Volume**
    - **Melt temperature**
    - **Back pressure**
    - **Melt leakage through the check ring during injection**.

Calculation of Discharge Density:
- Physically, we can determine discharge density if we know the **injected volume** and the **total injected weight**.
- In an existing process, the injected volume is determined using the **cylinder equation**:
    - Volume = Area × Length
    - Area = (Screw diameter)² × π/4
    - = Recovery position – Cushion position

Obtaining Discharge Density:
- In an existing process, you can obtain the **Recovery Position** and **Cushion Position** by navigating through the control pages.
- The **injection weight** is obtained by weighing the molded parts plus the runner (if Length present).
- Finally, discharge density is calculated by dividing mass by volume:

Discharge Density (g/cm^3) =

$$\frac{\textbf{[Total Injection Weight]}}{\textbf{[(Screw diameter)}^2 \times \pi/4\textbf{]} \times \textbf{[Recovery Position – Cushion Position]}}$$

What is the procedure for measuring melt temperature?

Variations in melt temperature can impact the molded product, including thermal dimensions (according to Universal Molding™) and plastic morphology. For this reason, it is crucial to verify the temperature using a standardized method or protocol.

Here are the key points:

1. **Barrel Heat Zones**: These zones are used to obtain the most critical temperature: the melt temperature.
2. **Heating Bands**: The heating bands in the barrel zones, along with friction, are responsible for melting the material. Back pressure affects friction.
3. **Thermocouples in Heat Zones**: Thermocouples measure the heat of the metal, not the melt.

Procedure:

1. Ensure that the process has operated normally for at least ten cycles.
2. Preheat an instrument to 25°C below the desired temperature. Digital "mini-blowers" are an economical option for preheating.
3. Stop the process (e.g., switch to semi-automatic mode). Once the mold opens, remove the injection unit and purge the melted material. You can do this on a removable surface for easy access.
4. Adjust the instrument to maintain the peak temperature value. This eliminates subjectivity when searching for the melt stabilization temperature.
5. Immerse the instrument in the melt and agitate it. When you notice the temperature starting to drop, remove the instrument and record the peak temperature obtained. Follow all safety rules.

Note:

- Use safety equipment such as uniforms, gloves, and goggles.
- Adapt this protocol to your processes and ensure that everyone measures melt temperature consistently.

Is it necessary to adjust process limits?

Injection molding process limits are necessary alarms and must always be set.

We have observed that many in the injection molding industry ignore the process limits. Some of these limits are the Maximum Injection Pressure limit, the Cushion Upper and Lower limits, and the Recovery Time limit. These alarms protect the equipment and the quality of the molded parts. Why are they ignored? Although some do it out of neglect, most operators ignore them because they do not know their benefits.

What would cause the Maximum Injection Pressure limit to be reached, and what could be its consequences?

It could be caused by a blocked cavity, either because a part was trapped in the cavity or a plugged gate. The injection unit is programmed to fill 100% of the mold; if for some reason a cavity is clogged, the control has no way of knowing and will continue injecting, causing high injection pressures. These high pressures could cause flash in the parts, melt leaks into the hot runner system or melt leaks between the mold sprue bushing and the nozzle tip.

What would cause the Cushion Low limit to be reached, and what could be its consequences?

Reaching the cushion low limit is an indicator that the melt is seeping somewhere; it is typically the result of a dirty or defective check ring, and it could be that it is leaking somewhere in the mold or between the sprue bushing and the nozzle tip. This melt leak condition, if ignored for prolonged times, could cause the melt to reach some of the heater bands of the barrel, could reach the wiring of the hot runner system, to sneak between mold actuators damaging the mold, in general substantial damages. In addition, if the low limit reaches zero cushion, totally empty injection unit, it will nullify the packaging stage and parts dimensions changes could be observed.

What would cause the Cushion High Limit to be reached, and what could be its consequences?

Reaching the cushion high limit is an indicator of a blocked cavity or clogged gate. The consequences could be over-packaging of parts, which could cause parts jamming in the cavity and part’s dimensional changes.

What would cause the Recovery Time Limit to be reached, and what could be its consequences?

Reaching the recovery time limit could be caused by a lack of material. The melt in front of the check ring is the one that pushes the screw towards the recovery position, and if the resin feed is interrupted, the recovery time will extend. Systems that integrate pigment dosing in the throat of the injection unit could observe variation in the color of the parts. Do not forget that If the recovery time extends the cooling time, it could trigger an alarm.

There are many consequences; use these limits appropriately, and you will enjoy their benefits and savings.

What is an appropriate upper and lower limit for Back Pressure?

Before providing an opinion, let's start with an overview. During plasticization, the screw rotates, and the melt is pumped through the check valve toward the front of the screw. The accumulated molten mass at the front of the screw pushes it to the plasticization position. Back pressure (BP) results from a controlled force opposing this displacement. The purpose of BP is to ensure consistently homogeneous melt (or improve additive dispersion in the melt) and to increase or decrease shear rate during loading.

Changing back pressure has multiple consequences. For example, when increased:

1. It enhances the mixing capacity of additives.
2. Degradation of sensitive materials and fiber breakage increase.
3. Screw and barrel wear intensifies.
4. Heat contribution from friction increases, or heat contribution from heater bands decreases.
5. The amount of molten mass increases; since molten thermoplastics are compressible, more plastic can be plasticized in the same volume. Consequently, more material is transferred to the mold during the injection stage.
6. Melt viscosity typically decreases due to heat from friction. As a result, in-machine rheology values change.

There are three typical options for BP limits:

1. Set BP without upper or lower limits.
2. Establish validated BP limits within a range.
3. No restrictions, allowing molders to decide.

Controlled industries (such as medical) usually work with one or two validated resins, often without regrind. Therefore, setting BP without upper or lower limits is justified. If, for any reason, they need to use BP limits (due to regrind usage), they should set tight, validated BP limits and maintain a consistent virgin-to-regrind ratio.

However, this doesn't mean that uncontrolled industries should have wide-open BP adjustments or leave it to the operator's discretion. In uncontrolled industries where multiple resin brands are used for the same product and the virgin/regrind ratio is not controlled, it doesn't make sense to establish narrow upper and lower BP limits.

Understand your material before setting BP limits. Ask yourself:

1. Is the material fiber-filled?
2. Does it degrade easily?
3. Does the material supplier change based on market prices?
4. Will additives like pigments or plasticizers be dosed?
5. Does the virgin/regrind ratio change due to warehouse limitations?

For uncontrolled industries, allowing operators to manipulate back pressure should not be the default option. Although melt index changes with material supplier and virgin/regrind ratio, a restricted upper and lower BP limit should be established and reviewed based on material changes.