

Universal Molding™

A Systematic Method for Injection Molding Optimization

Héctor Dilán
h_dilan@outlook.com

*Héctor Dilán carefully prepared this book and is believed to be correct. The book and the **Universal Molding™** discipline provide general guidance with no warranties of any kind. Héctor Dilán makes no warranties and disclaims any responsibility or liability of any kind for any loss or damage as a result of the use of or reliance upon such information.*

Any reproduction of any part of this book without the written permission of Héctor Dilán is prohibited.

5th Edition
Copyright Héctor Dilán, 2024

Dedication

I dedicate this book to my wife, Susan, for being the pillar that strengthens every day of my life, for helping me with the editing and for lending me family time to write this book.

Special Thanks

This book was written thanks to the motivation of friends from the plastic industry. Some of those friends and collaborators are Luis de Jesús, Carlos López, Iván Baigés, Gregorio Vélez, Runny Hernández, William Torres, Wally Cruz and Félix Colón. My friends, thank you for the collaboration and for sharing your knowledge with me.

Table of Contents

Figures	8
Preface	12
Prologue	13
I. Introduction.....	14
What is <i>Universal Molding™?</i>	15
Fundamentals of the Injection Molding Process	17
II. Injection Molding Process Parameters.....	25
Machine and Universal Parameters	26
Injection Parameters	27
Transfer Parameters	28
Hold Parameters	29
Gate Freeze	30
Cooling Parameters	31
Recovery Parameters	33
Press Movements	35
Questions.....	37
III. Process Graphs.....	40
Molding with Graphs	41
PVT Diagrams	50
Questions.....	52
IV. Plastic Morphology.....	54
Types of Plastics	55
Mechanical Properties	56
Common Materials and their Characteristics	58
Shrinkage.....	61
Some Experiments	62
Questions.....	67
V. Auxiliary Equipment.....	69
Material Drying	70
Extremely dry air.....	71
Drying temperature.....	71
Drying flow.....	72
Drying time.....	73
Drying equipment components.....	73
Drying hopper.....	74
Drying hopper size.....	75
Drying time.....	76
Bulk density.....	76
Resin loader.....	78
Dryer.....	78
Filter.....	79
Blower.....	79

Desiccant bed.....	80
Heater.....	81
Regeneration.....	81
Setting up drying systems.....	83
Questions.....	92
Blending and Material Handling.....	95
Direct and volumetric feeder.....	95
Gravimetric feeder.....	98
Pneumatic proportional valve.....	100
Central blending system.....	101
Questions.....	102
Controlling the Water Temperature to the Mold.....	103
Material Consumption.....	103
Heat removed.....	104
Water flow.....	106
Estimated Cooling Time.....	111
Temperature Control Units (TCU).....	113
TCU with direct cooling.....	114
TCU with direct cooling and heating.....	115
Questions.....	120
VI. Molding from the Desk.....	121
Press Calculations.....	122
Clamping force fundamentals.....	123
Projected area.....	124
Thin wall calculation.....	129
Forces resulting from side action mechanisms.....	132
Three-plate molds.....	136
Stack molds.....	137
Platen spacing on machines with tie bars.....	138
Platen spacing in tie-bar-less machines.....	139
Minimum and maximum openings.....	139
Ejector patterns.....	140
Questions.....	141
Injection Unit Calculations.....	142
Injection unit size.....	142
Nozzle tips and sprue bushings.....	144
Fountain flow effect.....	148
Density and specific density.....	150
Injection speed and injection flow.....	150
Barrel utilization.....	151
Recovery position.....	154
Discharge density.....	155
Recovery speed.....	157
Residence time.....	158
Transfer position.....	161
Summary.....	163
Temperature profiles.....	164

Procedure for measuring melt temperature.....	167
Intensification ratio (R_i).....	168
Machine labeling.....	169
Questions.....	170
VII. Machine Rheology.....	174
Plastic Melt Flow	175
Shear Stress, Viscosity and Shear Rate	175
Machine Rheology by Power	179
Machine Rheology by Viscosity	181
Approximated Rheology	185
Equation to Predict Injection Time	191
Questions.....	195
VIII. Determining Injection Speed.....	197
Laboratory I - Understanding Fill and Its Limitations, Determining Minimum Injection Time and Injection Pressure Limit	200
Laboratory II - Determination of Injection Time and Injection Speed	207
Laboratory III - Rheology Graph and Determination of Ideal Injection Time and Speed	212
Laboratory IV - Approximated Graph	213
Laboratory V. Injection Time Prediction	215
Questions.....	220
IX. Verifying Fill Balance.....	222
Effect of Injection Time on Fill Balance	224
Laboratory - Fill Balance	227
Thermal Imbalance	229
Imbalance in Molds with Cold Runners	230
Questions.....	231
X. Determining Hold Stage Parameters.....	233
Laboratory I - Determining Hold Pressure	235
Verification of injected volume percentage based on the screw's position.....	238
Laboratory II - Determining Hold Time	242
Hold Stage for Molds with Hot Runners and Gate Valves	244
Questions.....	246
XI. Determining Cooling Stage Parameters.....	249
Understanding Your Product	250
Cooling Time	251
Optimization Using the Mold Temperature	251
Procedure for measuring melt temperature.....	252
Other Parameters that Affect Cooling	258
How is a Cooling Experiment Organized?	260
Questions.....	262
XII. Process Limits.....	265
What would cause the maximum injection pressure limit to be reached, and what could be its consequences?	266

What would cause the cushion lower limit to be reached, and what could be its consequences?	266
What would cause the cushion upper limit to be reached, and what could be its consequences?	267
What would cause the recovery time limit to be reached, and what could be its consequences?	267
What is an appropriate upper and lower limit for back pressure?	267
Questions.....	269
Appendices.....	270
I - Troubleshooting	271
II - <i>Universal</i> Mold Data	277
III - <i>Universal MoldingTM</i> Equations	278
IV - General Procedures for <i>Universal MoldingTM</i>	287
V – English Terms in Spanish	291
VI - Spanish Terms in English	293
VII - Operational Costs	295
Bibliography	297
Answers	298
Index	303
Expert Opinions	305

Figures

<i>I-1. Injection molding machine rheology graph</i>	16
<i>I-2. Conventional and approximated rheology graph</i>	17
<i>I-3. The injection stage</i>	19
<i>I-4. The spaces that the plastic occupies in the mold</i>	20
<i>I-5. The check ring</i>	21
<i>I-6. Position of the check ring during recovery</i>	22
<i>I-7. The recovery stage</i>	24
<i>II-1. The injection stage</i>	27
<i>II-2. Walls with cold molds and extended cooling times</i>	32
<i>II-2a. Walls with hot molds and shortened cooling times</i>	32
<i>II-3. Backpressure</i>	33
<i>II-4. Decompression</i>	34
<i>III-1. Graph of ideal injection</i>	41
<i>III-2. Control zone graph of injection speed or flow</i>	43
<i>III-3. Graph of hold zone or pressure control</i>	44
<i>III-4. Graph of recovery zone</i>	46
<i>III-5. Graph with limited pressure</i>	47
<i>III-6. Graph illustrating premature transfer</i>	48
<i>III-7. Graph illustrating cushion equal to zero</i>	49
<i>III-8. Graph illustrating unachieved programmed injection speed</i>	49
<i>III-9. PVT (Pressure, Specific Volume and Temperature) diagram</i>	50
<i>III-10. PVT diagram with molding stages</i>	51
<i>IV-1. Illustration representing amorphous and semi-crystalline molecular organization</i>	
<i>IV-2. Graph illustrating stiffness versus temperature in amorphous materials</i>	55
<i>IV-3. Graph illustrating stiffness versus temperature in semi-crystalline materials</i>	56
<i>IV-4. Overlaid graphs of stiffness versus temperature of semi-crystalline and amorphous materials</i>	57
<i>IV-5. Common amorphous and semi-crystalline materials</i>	58
<i>IV-6. List of mechanical characteristics of amorphous and semi-crystalline materials</i>	60
<i>IV-7. List of process characteristics of amorphous and semi-crystalline materials</i>	61
<i>IV-8. Amorphous material shrinkage</i>	61
<i>IV-9. Semi-crystalline material shrinkage</i>	62
<i>IV-10. The effect of hold time on the part weight in amorphous and semi-crystalline materials</i>	63
<i>IV-11. The effect of hold pressure on the part weight of amorphous and semi-crystalline materials</i>	64
<i>IV-12. The effect of injection pressure on injection time with amorphous and semi-crystalline materials</i>	65
<i>V-1. The mechanics of drying thermoplastics</i>	72
<i>V-2. Drying hopper and dryer</i>	74
<i>V-3. Drying hopper</i>	75
<i>V-4. Volume/residence time</i>	76
<i>V-5. Drying circuit</i>	79
<i>V-6. Regeneration stage</i>	82

<i>V-7. Portable unit</i>	84
<i>V-8. Hopper above the extruder</i>	84
<i>V-9. Integrated unit</i>	85
<i>V-10. Central drying system</i>	85
<i>V-11. Table of materials and their drying parameters</i>	87
<i>V-12. Table of materials and their bulk densities</i>	89
<i>V-13. Dry air flow required for each lb/hr of material consumption</i>	91
<i>V-14. Additive feeder</i>	96
<i>V-15. Graph of speed adjustment vs. feeding rate</i>	97
<i>V-16. Two feeders mounted in one system</i>	98
<i>V-17. Gravimetric system</i>	99
<i>V-18. Pneumatic proportional valve</i>	100
<i>V-19. Central blending system</i>	101
<i>V-20. Table of energy required for some materials</i>	104
<i>V-21. Table of thermal load of some materials</i>	105
<i>V-22. Water flow through the mold and Delta T</i>	107
<i>V-23. Water hoses connected to the mold in series and in parallel</i>	107
<i>V-24. Temperature between cavities</i>	108
<i>V-25. Table of constants for cooling time equation</i>	113
<i>V-26. Inlet and outlet water temperature and pressure</i>	114
<i>V-27. Diagram of direct cooling components</i>	115
<i>V-28. Diagram of direct cooling and heating components</i>	116
<i>V-29. Three machines sharing a chiller</i>	118
<i>V-30. Three machines sharing a chiller, with a stopped machine</i>	119
<i>VI-1. Table of pressure factors for some materials</i>	123
<i>VI-2. Projected area of a sphere</i>	124
<i>VI-3. Projected area of a cube</i>	124
<i>VI-4. Projected area of a cup</i>	125
<i>VI-5. Nylon spools and runners with and without parts</i>	126
<i>VI-6. Projected area of one of the 12 spools and its runner</i>	127
<i>VI-7. Area of the runner</i>	127
<i>VI-8. Example of an L-shaped part</i>	129
<i>VI-9. Flow path of a spool</i>	130
<i>VI-10. Thickness (T)</i>	131
<i>VI-11. Table of thin wall criteria</i>	131
<i>VI-12. Linear interpolation of the pressure factor</i>	132
<i>VI-13. Side action mechanisms (sliders) driven by clamping force</i>	133
<i>VI-14. Measurements to calculate the projected area</i>	133
<i>VI-15. Lateral force per slider</i>	134
<i>VI-16. Resulting force</i>	135
<i>VI-17. Three-plate mold</i>	136
<i>VI-18. Stack mold</i>	137
<i>VI-19. Stack mold force vectors</i>	137
<i>VI-20. Platen spaces with tie bars</i>	138
<i>VI-21. Platen spaces on tie-bar-less machines</i>	139
<i>VI-22. Minimum press opening</i>	139
<i>VI-23. Maximum press opening</i>	140
<i>VI-24. Ejector pattern</i>	140

VI-25. Maximum recovery	142
VI-26. The nozzle tip and sprue bushing	144
VI-27. Diameters and radii of the nozzle tip and sprue bushing	145
VI-28. Defect caused by stress concentration at the edge of the nozzle	146
VI-29. Defect caused by a sprue stuck in the nozzle	147
VI-30. Another defect caused by a sprue stuck in the nozzle	147
VI-31. Thermoplastic melt flow in a mold	148
VI-32. The fountain flow effect	149
VI-33. Example of fountain flow effect using a piece of napkin	149
VI-34. Fill positions	153
VI-35. Residence volume	158
VI-36. Table showing example of residence T_r	159
VI-37. Table of residence time, according to % of utilization	160
VI-38. Table showing % of utilization according to industry type	160
VI-39. Transfer position	161
VI-40. Tables showing transfer position criteria	162
VI-41. Screw positions	164
VI-42. The barrel's heat zones	164
VI-43. Temperature profiles	166
VI-44. Intensification ratio	168
VII-1. Orientation of molecules in a melt flow	175
VII-2. Shear stress	176
VII-3. Half of the melt flow	176
VII-4. Velocity vectors in a melt flow	177
VII-5. Newtonian and pseudoplastic melt flow	178
VII-6. Graph of machine rheology by power	180
VII-7. Graph of injection time versus peak power	180
VII-8. Zone where the change in injection time is minimal or where the power stopped contributing	181
VII-9. Zone where the change in relative viscosity is minimal	183
VII-10. Readings of transfer pressure and injection time	184
VII-11. Graph of linear behavior between injection flow and peak power	185
VII-12. Graph of linear behavior between relative viscosity and injection time	186
VII-13. Example of injection times and transfer pressures	186
VII-14. Examples of rheology values	187
VII-15. Graph of the linear effect between average injection flow and peak power	187
VII-16. Graph of the linear effect between relative viscosity and injection time	188
VII-17. Table including 8 equidistant injection times	189
VII-18. Graph of approximated rheology by power	190
VII-19. Graph of approximated rheology using relative viscosity and shear rate	190
VII-20. Superimposed graphs of complete and approximated reology	191
VII-21. Graph of standardized rheology in %	192
VII-22. Injection time for conventional molding industries	193
VII-23. Injection time for industries that mold friction-sensitive materials	193
VII-24. Injection time for high-volume injection industries	193
VIII-1. Flow chart of Universal Molding™ events	198
VIII-2. Tables to select the injection-to-hold transfer position	202

<i>VIII-3. Determination of minimum injection time and maximum injection pressure (example)</i>	204
<i>VIII-4. Effect of injection time on parts fill</i>	205
<i>VIII-5. Rheology table headers</i>	208
<i>VIII-6. Table of approximated rheology by power</i>	209
<i>VIII-7. Table showing conventional rheology by power</i>	210
<i>VIII-8. Example of a table and graph of conventional rheology by power</i>	212
<i>VIII-9. Example of a table of approximated rheology by power</i>	213
<i>VIII-10. Graph with linear equation between peak power and average injection flow</i>	214
<i>VIII-11. Table and graph of approximated rheology by power</i>	215
<i>VIII-12. Table of approximated rheology by power</i>	216
<i>VIII-13. Rheology graph indicating injection times by industry type</i>	217
<i>IX-1. Multi-cavity mold with unbalanced fill</i>	223
<i>IX-2. Graph of the effect of injection speed on the fill of incomplete parts</i>	224
<i>IX-3. Fast and slow fill</i>	224
<i>IX-4. Example of flow sequence of the runner</i>	225
<i>IX-5. Example of slow fill</i>	226
<i>IX-6. Example of fast fill</i>	226
<i>IX-7. Example of incomplete fill</i>	227
<i>IX-8. Parts separated from the runner</i>	228
<i>IX-9. Table with incomplete cavity weights and corresponding fill deviations</i>	228
<i>IX-10. Column plot with % of cavity fill deviation</i>	229
<i>X-1. Graph of the effect of hold pressure on the weight of the parts</i>	234
<i>X-2. Extended cooling time</i>	235
<i>X-3. Table of the effect of hold pressure on the weight of the parts</i>	236
<i>X-4. Graph of the effect of hold pressure on the weight of the parts</i>	237
<i>X-5. Graph indicating hold pressure range</i>	237
<i>X-6. Determination of hold pressure with the cushion's position</i>	238
<i>X-7. Graph indicating excessive hold pressure</i>	240
<i>X-8. Correction when cushion is zero</i>	241
<i>X-9. Extended cooling time</i>	242
<i>X-10. Table of the weight of the parts and their respective hold and cooling times</i>	243
<i>X-11. Graph of the effect of hold time on the weight of the parts</i>	244
<i>X-12. Mechanical characteristics of materials which should be considered during hold</i>	
<i>XI-1. Mold temperature optimization experiment table</i>	253
<i>XI-2. Example of parts of a two-cavity mold</i>	255
<i>XI-3. Table with effects of mold temperature on critical measurement</i>	256
<i>XI-4. Graph of mold temperature effect on critical measurements</i>	256
<i>XI-5. Graph showing linear equations using temperatures and measurements of a two-cavity mold</i>	257
<i>XI-6. Table with temperature limits at critical measurements of two parts</i>	258
<i>XI-7. Table of cooling parameters (temperature and time), with their combinations and their repetitions</i>	260

Preface

Injection molding was, for many years, a space occupied by the experienced masters of the plastics industry. Fortunately, today there are several techniques that accelerate learning and mastery of injection molding.

We have experimented with numerous molders of all calibers. Among us we have never found two that use identical molding techniques. Not only are the molding techniques different, but language, order of execution, definitions, parameter usage, equipment identification, procedures, and communication are not equal. These are the reasons that motivate the writing of this book.

Universal MoldingTM (abbreviated ***MUTM***) aims to unify molding styles, use valid process definitions, use Universal language, and standardize clear and representative procedures for all stages of injection molding.

This book is not limited to just newcomers to the industry; it is also recommended for experienced molders who wish to standardize and increase the number of competent molders in their sector. ***Universal MoldingTM: Systematic Injection Molding Optimization Method (MUTM)*** is for anyone who wants to learn systematically and effectively about injection molding. ***MUTM*** guides the molder to determine the Universal parameters characteristic of the mold, regardless of the injection machine used. There is a concern that affects a minority in the plastic industry, artificial intelligence. We see it as a tool that will help address the lack of standardization and empirical approaches in process parameter optimization. The integration of artificial intelligence in ***MUTM*** considers the fundamental principles of standardization, rapid learning, efficient processes, and significant benefits. We must all strive to improve, accept, and learn new methodologies and technologies that strengthen continuous growth.

Prologue

What is the definition of an “optimal” injection polymer molding process? In reality, we do not establish an optimal process, but instead define an optimal operational window around the values of certain molding parameters that an optimization laboratory leads us to establish and define as ‘nominal’. Using the tools provided by the internet we can find countless definitions and/or methods on what is an optimal process, how to establish it, and even how to monitor it. These definitions, in short, establish that an optimal polymer molding process is *“the result of a particular combination of molding parameter values... inside an operational window... able to consistently produce a plastic part that meets all cosmetic, dimensional and functional design requirements... in the shortest machine time possible.”* This, for any combination of raw material, mold design and molding cell (i.e. injection machine, dryer, temperature controls, etc.). Similarly, that optimal process *cannot be* if it goes against the ability of the mold, or even the molding machine, in order to repeat the same cycle of behavior infinitely.

The author of the book has devoted much of his professional life to systematically identifying two things. First, how to establish that optimal operational window in a molding process; and second, how to make that optimal process window for any combination of raw material, mold and molding equipment the same regardless of who runs it or where the optimization lab is performed. That is why *“Universal”* is in the title of this book. It is the intention of the author, with the tools presented in this, his book, to make the Process Engineer able to identify the minimum requirements with which each of the equipment and utilities that make up the molding cell must meet. Once the equipment is successfully selected, it provides the tools for the selected equipment to become the foundation on which, in a complete way, the Process Engineer can develop the laboratory and establish the design experiments that will shape that combination of ‘nominal’ parameters around which will define the limits of that optimal operation window being validated. This....*in the shortest machine time possible.*

Felix Colón Ortiz
Injection Molding Process & Tooling Engineering Professional

I. Introduction

- What is *Universal MoldingTM (MUTM)*?
- Fundamentals of the Injection Molding Process

In this section we want the readers to familiarize themselves with the terminology, emphasizing those parameters that are most significant to the process and establishing the language used in this book.

What is *Universal MoldingTM*?

Universal MoldingTM (MUTM) is an injection molding process optimization discipline. It was developed with the collaboration of the Caribbean plastic industry and the academia (professor and students) from the University of Puerto Rico, Mayagüez Campus (UPR RUM).

MUTM is a *discipline* that emphasizes the maximization of resources and focuses on the quality of the product, utilizing process optimization methods proven by means of organized and scientifically backed molding techniques. This tecno-scientific background increases efficiency, decreases product cost, and shortens manufacturing cycles.

MUTM is a *common language* used by molders to eliminate terminology confusion. The equipment is labelled with a language that represents their capacities. It is this language of **Universal** process parameters that simplifies the transference of processes between machines. It is a language that defines a product and its utilization.

MUTM is an *organizing committee* (or **Universal** committee). It is a chosen group that promotes that discipline. It is a **Universal** committee represented by all departments of the **Universal** factory. It is represented by the Production, Quality Control, Equipment Maintenance, Mold Maintenance, Engineering and Sales departments.

MUTM is an *endless discipline* that never ceases to grow or improve. The **Universal** committee has the responsibility to evaluate and unanimously adopt procedures that improve the existing ones.

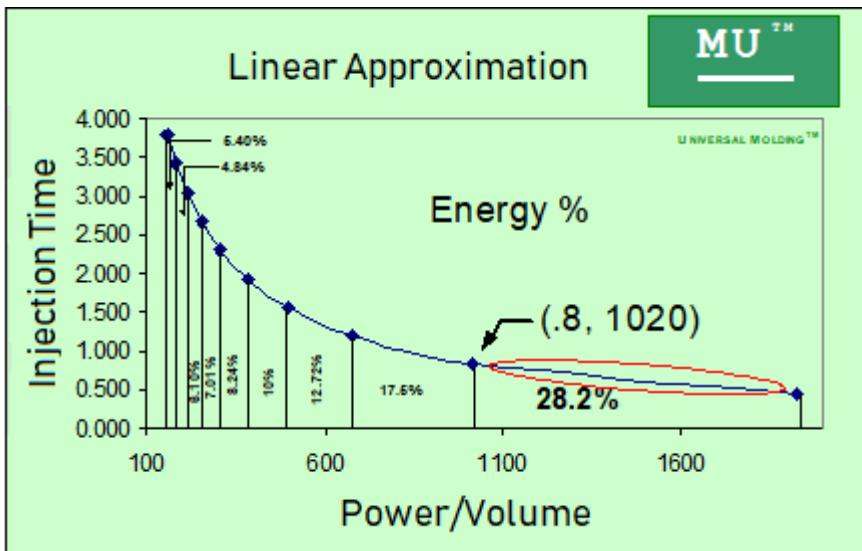
MUTM is *based on precise and representative process procedures*. At each stage, a procedure is followed to determine the parameters, either through linear equations or, in many cases, with a nonlinear component generated by artificial intelligence (AI).

MUTM is *maximizing the utilization* of the machinery. It is determining the appropriate equipment and its optimal process parameters.

Some of the techniques used are:

- 1- Injection machine rheology. This is an effective, proven technique used to determine injection time. Using a graph, it shows the effect of the injection time versus energy per unit of volume.

In the following graph, the area below the curve demonstrates the percentage of energy consumed for each decrement in injection time.



I-1. Injection molding machine rheology graph

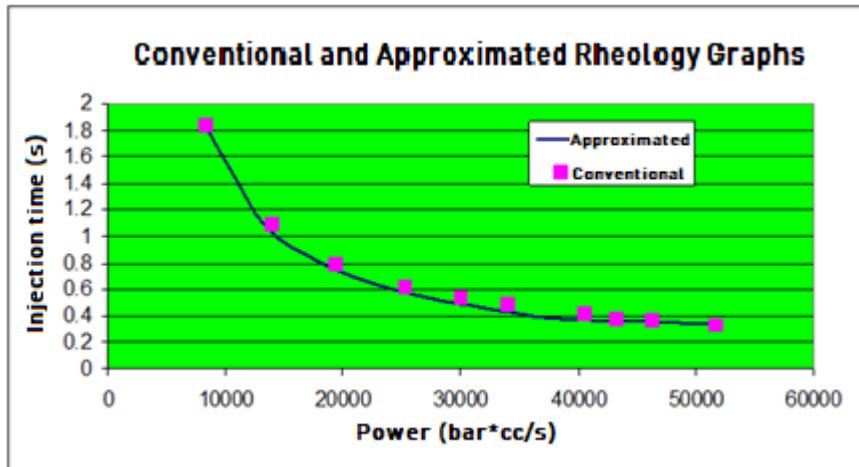
Note that the power consumption increases as the injection time decreases. This graph shows that the power required at higher injection rates is considerably high, or the power consumed by the injection unit is more significant at lower injection times. The idea is to select an injection time in the zone in which the time stops contributing with an increase in power.

Later we will explain how to develop and utilize this injection molding rheology graph.

- 2- Approximated rheology. Developing a rheology laboratory with an injection machine consumes time and resources. With approximated

rheology, a mathematical prediction technique, the laboratory can be achieved in less than a third of the time.

The following graph of injection time versus power by volume compares the two methods: conventional machine rheology and the approximated method.



I-2. Conventional and approximated rheology graph

Both methods conceptually function in the same way. The difference is that using the approximated method consumes less time and resources.

Again, the development of these graphs will be explained in later chapters.

Fundamentals of the Injection Molding Process

The basic stages of the injection molding process are:

- injection
- changeover or transfer
- hold
- gate freeze
- cooling
- recovery

Each stage has a function and a specific result. Understand each one of these stages thoroughly since their descriptions will be continually referenced.

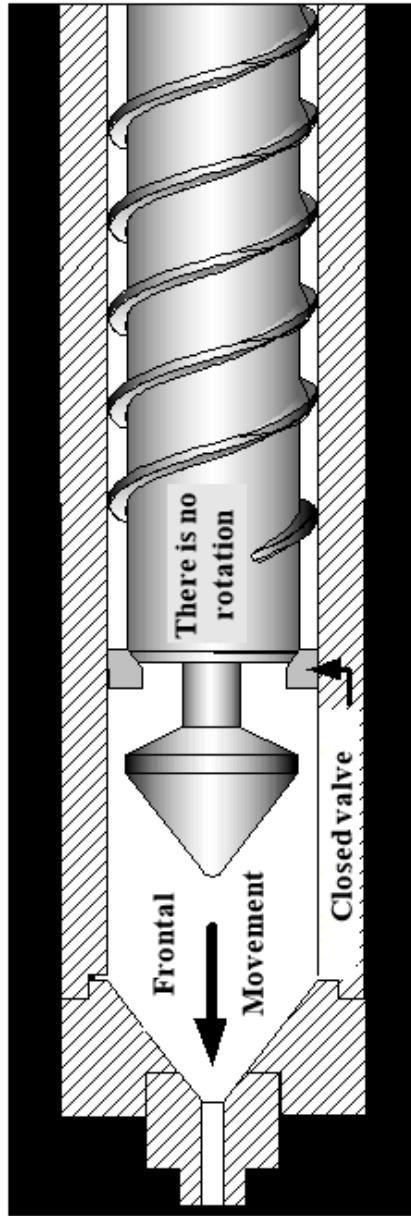
Injection - In this stage the mold cavities and runner are filled close to 95% of their total volume, and the screw acts like a piston that transfers melt from the injection unit to the mold. Here is where you program a velocity or injection flow rate that guarantees the best melt properties. These properties could be parts without burns, no flowlines, no degradation, minimal stress concentration, etc. When the hot melt enters the mold, it is met with cold walls and rapidly densifies until it solidifies. The slow fill increases densification or viscosity and, consequently, it could make filling the mold difficult and may even cause the melt to solidify prematurely before the mold has been filled. In this stage the injection time, as well as the injection pressure, are results and not control parameters. Do not confuse these with *injection pressure limit* or with *injection time limit*, which are limits that are programmed to protect the tooling and the equipment. This stage is known as the injection speed control stage.

Transfer (changeover) – This is what determines the end of the injection stage. Once the injection unit has filled the mold **close to 95%**, the injection stage ends, and the hold stage begins. The injection unit comes with a linear encoder that measures the injection screw displacement, which is how the injection unit knows when the melt is close to filling 95% of the mold. Avoid trying to fill the mold 100% in the injection stage. Let's see some of the reasons why:

- It could cause flash on the molded parts. What stops the screw is the melt in front of the injection unit; trying to stop at exactly 100% without opening the mold would be difficult.
- At a high speed, trying to fill a mold to 100% could create a bounceback effect on the screw. Plastic melt is compressible and during injection it is placed under pressure. This pressurized melt can act like a compressed spring, pushing the injection backwards and causing a suckback effect that pulls back part of the melt that was injected.
- Another reason that it should not be done is because of material shrinkage. Melt occupies more space than solidified material.

Once the melt enters the mold it will cool, gradually shrinking and leaving space for more material.

Note: Some molds present an extreme difficulty in filling, for example, nylon ties which are long and thin, or micro-molding applications with narrow and awkward spaces for filling. In these cases, a filling percentage higher than 95% may be required.



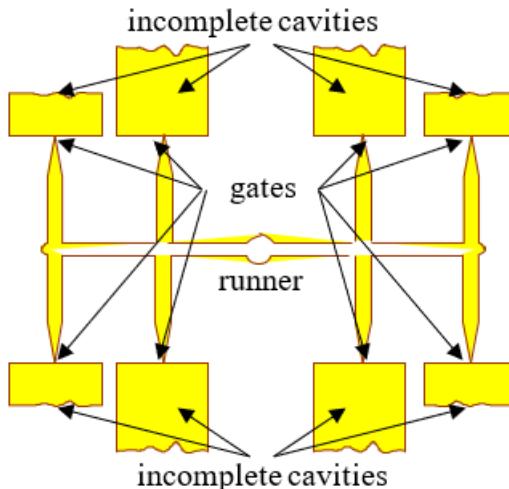
During the injection stage the screw acts like a piston injecting a fluid. The melt is maintained at the front of the screw and the valve keeps the melt from returning to the screw. The displacement of the screw begins at the recovery position and will continue until the changeover to hold.

I-3. The injection stage

Hold - In this stage the screw continues to move like a piston, forcing more melt into the cavities until it fills the portion that was not filled in the injection stage. Without opening the mold, the injection unit compresses the melt, packing more material into the mold until the cavities are completely filled. Here the molder adjusts the compacting pressure.

During this stage we achieve the proper weight for the molded parts, or what we **Universal** Molders call *mass dimensions*. The mass dimensions are those that are a function only of the quantity of material and should not be confused with the dimensions that are due to the effect of material shrinkage. Shrinkage is controlled during the cooling stage. As indicated previously, during the hold stage we only control the mass dimensions, the dimensions that are a function of the quantity of material.

Gate Freeze – During the hold stage, the parts are pressurized until the material in the gates solidifies, creating a seal that keeps the melt inside the cavities. Let's look at the spaces that the plastic occupies in the mold.



I-4. The spaces that the plastic occupies in the mold

A gate is a small opening through which the melt enters the cavities. The melt enters the mold through a sprue and travels through the runner until it reaches the cavity gates. The melt is forced through the narrow

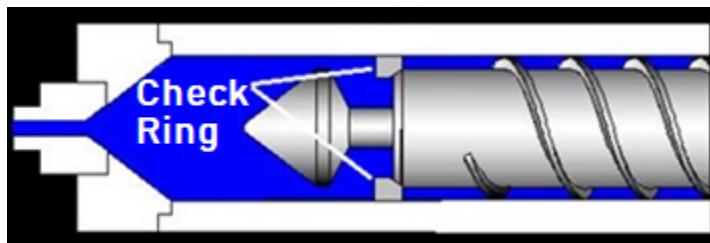
openings of the gates until the cavities are filled. The plastic is held inside the cavities until the gates solidify. It is important to understand:

- If you remove the hold pressure prematurely, the melt will return to the runner and even to the injection unit.
- If the hold time is more than required, the molder will end up “molding runners”.

In some molds with hot runners, the melt never solidifies and is integrated as part of the filling for the next parts. The goal of this type of mold is to reduce the waste of material from the sprue. However, even in this case, the gates on the cavities must solidify before releasing the hold pressure.

In other molds, in addition to having hot runners, valves are integrated into the gates. These gate valves remain open during filling and close once the holding is complete.

The screw acts as a piston thanks to the check ring that floats between the screw and the screw tip. During injection, this check ring moves against the screw, creating a seal and keeping the melt from returning to the screw.



I-5. The check ring

During injection, the pressure in front of the check ring is greater than the pressure on the screw side, causing the check ring to move against the screw to create a seal.

There exist some screws that do not have a check ring. Rigid PVC material is very sensitive to the friction of the melt against the check ring, and it is common to see that this type of system does not use any check

ring. Instead, these screws come equipped with an anti-rotation mechanism so that they will not turn as a consequence of excessive melt pressure.

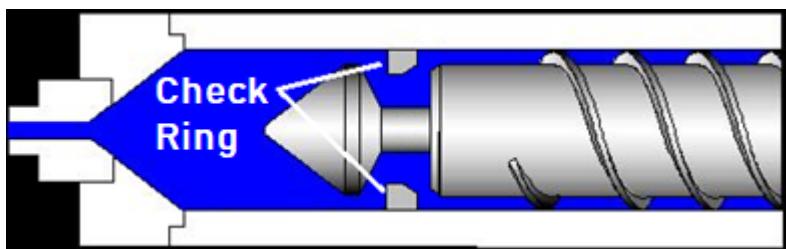
Cooling – In this stage, heat is removed from the parts, until they can be easily demolded with acceptable thermal dimensions. Thermal dimensions are dimensions that are a function of shrinkage and not of the quantity of packed mass. The molecules of thermoplastic melt are in continuous movement; as they cool, they look for conformity and accommodate themselves to occupy less space. The objective is to paralyze the molecular activity and manipulate this shrinkage to our advantage.

This means:

- Cold molds and extended cooling times result in parts with thicker walls.
- Hot molds and short cooling times result in parts with thinner walls.

Thermal dimensions and some mechanical properties are a function of how quickly the heat is being removed from the parts. These mechanical properties could include rigidity, translucence, crystallinity, etc. Later on, we will explain how thermal dimensions are a function of cooling time and mold temperature.

Recovery - In this stage, the screw reloads material for the next shot. The main goal of this stage is to produce a homogeneous melt. During recovery the check ring moves away from the screw, which allows the melt to flow to the front of the screw as the screw turns.



I-6. Position of the check ring during recovery

The melt that accumulates in front of the screw is what pushes the screw backwards.

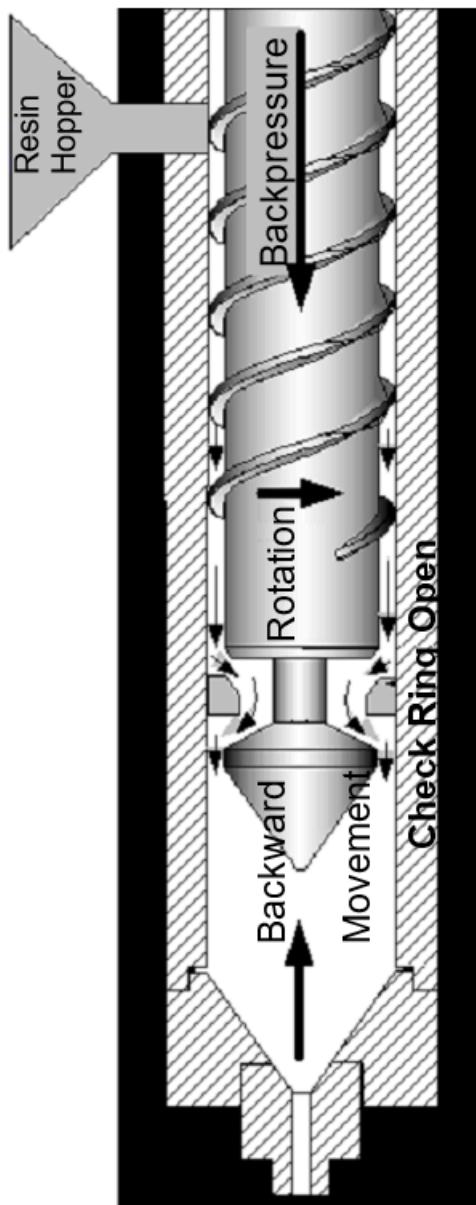
Recovery occurs at the same time as the cooling stage. Under normal circumstances recovery ends before the cooling stage ends and, if the cooling stage ends first, permission to open the mold is denied by the machine's controls. Under these circumstances, where the permission to open the mold has been denied and if no alarms exist that would cause the process to stop, the cooling time will be extended, altering the thermal dimensions.

Imagine what would happen if the mold opened during the recovery stage. The melt would drool into the mold. During recovery, the plastic is pressurized and the mechanism that holds the melt in place is the filled mold. As a rule, recovery should end close to a second before cooling. Permission to open the mold during recovery can only occur if the injection unit has been equipped with a valve on the nozzle.

It is important to know that the injection unit utilizes two sources of heat to melt the plastic: the heating bands on the barrel and friction. Generally, 50% of the heat comes from the heater bands and the remaining 50% comes from the friction of the plastic moving inside the barrel.

Later we will discuss in more detail the parameters that govern recovery, which are recovery speed, recovery position, backpressure, decompression and melt temperature.

Mold Movement – During this stage we demold the parts. Once the cooling stage has ended, the sequence is: the mold opens, if cores exist they will disarm in order to liberate the parts, the parts are ejected, the cores are relocated into the mold, the mold begins to close, the mold protection system is activated and, if the mold protection does not detect any issues, the injection machine reaches full closure force, and a new cycle begins.



In the recovery stage the screw rotates, moving the melt to the front of the barrel. The check ring moves, allowing the flow of melt. That melt accumulation in the front is responsible for pushing the screw backwards. The backpressure opposes the free movement of the screw, resulting in shear friction on the material, which, in turn, generates heat and contributes to the melting process.

I-7. The recovery stage

II. Injection Molding Process Parameters

- **Machine and Universal Parameters**
- **Injection Parameters**
- **Transfer Position**
- **Hold Parameters**
- **Gate Freeze**
- **Cooling Parameters**
- **Recovery Parameters**
- **Mold Movement**

Machine and Universal Parameters

Machine parameters are those that can be programmed into the machine control. These might not exactly describe the dynamics of the melt before it enters the mold; these could be:

- injection speed
- position of the screw
- the injection's hydraulic pressure
- revolutions per minute during recovery

Universal parameters are those values that the melt experiences before entering the mold, which could be:

- injection time
- melt volume
- plastic (or melt) injection pressure
- recovery time

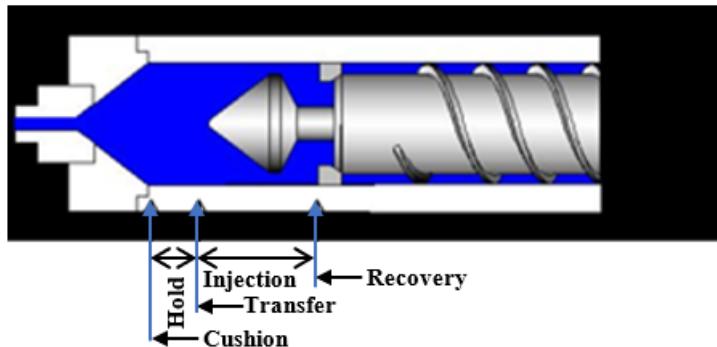
Both sets of parameters, **Universal** and machine, must be understood; of course, the parameters that should make more sense for **Universal** molders are **Universal** parameters. A **Universal** parameter better describes the physical function of filling a mold. Let's look at some examples:

- The ideal filling time of a mold is independent of the injection unit diameter and of its corresponding injection speed.
- Hydraulic injection pressure is distinct from and less than the melt pressure or the plastic pressure.
- The screw position represents a volume, a cubic quantity of melt that will occupy a space inside the mold after being injected. This melt volume can only be related to the injection screw's position when the screw's diameter is provided.

Not all injection molding machine controls utilize **Universal** parameters; even so, many manufacturers have recognized this as necessary and are starting to convert. In some newer machines, it is common to see that the control has the option to work with either **Universal** values (sometimes called "absolute") or machine values.

Injection Parameters

We previously mentioned that, during the injection stage, we want to fill the mold close to 95% of its total volume.



II-1. The injection stage

In this stage, filling is achieved with a quick injection flow that provides the best melt properties.

The injection machine parameter is:

- injection speed (in/s or mm/s)

Universal injection parameters are:

- injection time (seconds)
- injection flow (in^3/s or mm^3/s)

We the ***Universal*** molders utilize only one injection speed; there are molds that require speed profiles. For example, there are molds whose fill is not balanced, like family molds which create distinct parts at one time. Remember that many existing molds were manufactured years ago and are probably not in the best condition.

We have found that some molders have the habit of using a slow speed at the end of the injection stage in order to avoid screw bounceback. This situation is corrected by properly programming the transfer and recovery positions. Later we will explain how to determine those positions.

Again, we do not control injection pressure nor time in the injection stage. They are not controlled; however, during the injection stage we do program pressure and time *limits*. These limits should not be reached during the injection stage since their purpose is to protect the mold and the press.

Remember:

- The injection stage is known as the speed control stage.
- Do not try to fill the mold 100% during the injection stage; filling close to 95% is the goal.
- During injection, the screw acts like a piston.
- During injection, the check ring moves against the injection screw seat, creating a seal so that melt cannot return to the screw.

Transfer Parameters

Transfer is what determines the end the injection stage. Once the injection unit fills close to 95%, the injection stage ends and the next stage, hold, begins.

Injection machines can initiate transfer in four ways:

- time
- position
- pressure
- cavity sensor

Let us review each of these four ways:

- Transfer by time measures the time from the beginning of injection until reaching the programmed transfer time.
- Transfer by position measures displacement from the beginning of injection until reaching the programmed transfer position.
- Transfer by pressure monitors the actual pressure until reaching the programmed pressure.
- Transfer by means of a temperature or pressure sensor placed near the end of a cavity or cavities in the mold. When the melt enters the cavity and reaches the sensor set adjustment, the sensor sends a signal to the press' controls, indicating to transfer.

Universal molders prefer to transfer by position, since this best guarantees the injection volume required by the mold.

Some molders believe that the most reliable transfer method is by cavity sensors. Nowadays, few molds possess this technology. This is why we recommend transfer by position. Although it had gained some popularity, the investment in this type of cavity sensor technology cannot be justified for many applications. If your mold is equipped with them, then transfer by cavity sensor.

Machine transfer parameter:

- transfer by position (inches or millimeters)

Universal transfer parameters:

- transfer volume (cubic inches or cubic millimeters)
- transfer by cavity sensor(s) (temperature or pressure)

Hold Parameters

In this stage, the screw continues acting as a piston, compressing additional melt into the cavities until filling the remaining space that was not filled during the injection stage.

The hold stage is known as the pressure control stage. During this stage, we control the pressure in order to achieve the proper weight of the molded parts, which **Universal** molders call “mass dimensions”. Remember that mass dimensions are only a function of the quantity of material and should not be confused with dimensions that are an effect of shrinkage. The objective is to manipulate the hold pressure to guarantee dimensions that are a function of the quantity of material.

Machine hold parameter:

- hydraulic hold pressure (psi or bars)

Universal hold parameter:

- plastic (melt) hold pressure (psi or bars)

Remember:

- Cavity pressure is also a **Universal** parameter.

- Shrinkage will be controlled mainly during the cooling stage.
- During hold we only control mass dimensions, the dimensions that are a function of the quantity of material.

Gate Freeze

Once the parts are packed, the melt is held in place until the material in the gates solidifies, creating seals that trap the melt within the cavities.

In some molds with hot runners, gate valves are integrated into the runners. These valves remain open during filling and close when holding is complete. The signal that would correspond to the hold time is what activates the closure of the valve. The main objective of these valves is to achieve an aesthetic finish at the filling point of the cavities. However, although this type of valve usually reduces hold time, it does not necessarily improve the overall process cycle. This is because the time saved during hold is offset by a longer cooling time. Additionally, caution is important, as excessive holding time could cause the material to solidify at the filling point and damage where the valve sits.

The parameter that controls this gate freeze is hold time, whose units are seconds. Hold time is both a ***Universal*** and machine parameter.

It is possible to find controllers that divide hold into two parts, pack and hold. These separate the stages of pressure control and gate freeze. In addition to pressure and time, they consider hold speed.

In ***Universal Molding™*** we do not divide this stage into two parts. We believe that hold time starts when injection finishes and ends after the gates have solidified.

In ***Universal Molding™*** we do not consider hold speed. For the ***Universal*** molder, hold is the stage used to control pressure and speed is a result, since it is physically impossible to simultaneously control both pressure and melt velocity. One is the result of the other:

- if you control pressure, the result will be melt speed and
- if you control speed, the result will be melt pressure.

If you find a machine that has both options, pack and hold, turn one of them off (if possible) or adjust the secondary stage to zero.

We recommend only one pressure, though there are some older molds that still require a second pressure. For example, a mold could require a first hold pressure in order to guarantee mass dimensions and a second pressure to guarantee the demolding of the runner. Normally, the runner is the last part to solidify, and even though they are not what we are molding, the process may require the runners to be packed for them to demold properly.

Cushion

The cushion is the small amount of plastic that always remains in front of the screw after hold ends and should never be equal to zero. If the position of the screw reaches zero it will invalidate the hold stage and, consequently, there will be no control over mass dimensions. The cushion should always exist.

Remember:

- If you prematurely remove the hold pressure, the melt will return to the runner and possibly go as far as into the injection unit.
- If hold time extends too long, the molder ends up “molding runners”.
- During the hold stage, the check ring moves against the screw seat, sealing it and ensuring that the melt doesn’t return to the screw.

Cooling

In this stage, we remove heat from the parts until we obtain parts that can be demolded with acceptable thermal dimensions. The idea is to manipulate the removal of heat from the melt, in order to control the thermal dimensions. Remember that rapid cooling results in thicker walls and slow cooling results in thinner walls.

Cooling Parameters

- temperature of the mold ($^{\circ}\text{F}$ o $^{\circ}\text{C}$)
- cooling time (seconds)

These are both ***Universal*** and machine parameters.

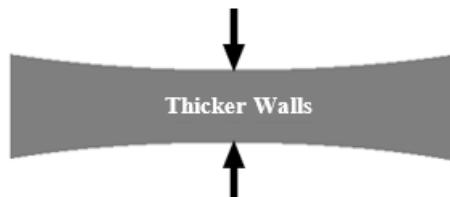
Familiarize yourself with your machine's controls since some controls include hold time inside of cooling time.

Be careful with this stage, extensive cooling times can result in costly processes. The idea is to manipulate both parameters, temperature and cooling time, in order to obtain the optimal thermal dimensions. Also, a change in mold temperature could affect hold time, since any change in the way heat is removed from the melt will affect the time it takes for the gate to freeze.

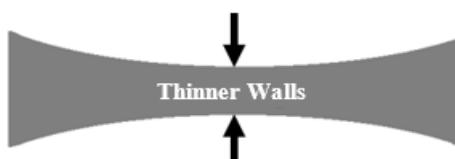
The mold temperature is controlled by changing the temperature of the water that cools the molds and sometimes by modifying its flow. It is important to understand that the temperature of the water that enters the mold and the temperature of the mold's metal are distinct.

Remember:

- Cold molds and extended cooling times will result in thicker walls.
- Hot molds and shortened cooling times will result in thinner walls.



II-2. Walls with cold molds and extended cooling times



II-2a. Walls with hot molds and shortened cooling times

- Some mechanical properties, like rigidity, translucency, and crystallinity, can also be functions of the rate at which the heat is removed.
- Thermal dimensions are a function of shrinkage and not a function of the amount of the packed mass.

Recovery Parameters

The main objective of recovery is to consistently produce a homogenous melt. During recovery the check ring moves to the front, permitting melt to pass in front of the screw.

Remember that recovery happens at the same time as the cooling stage. Under normal conditions, recovery ends before cooling does; if the cooling finishes first, the control will not allow the mold to open. This condition will extend the cooling time and consequently, will affect the thermal dimensions.

Machine recovery parameters:

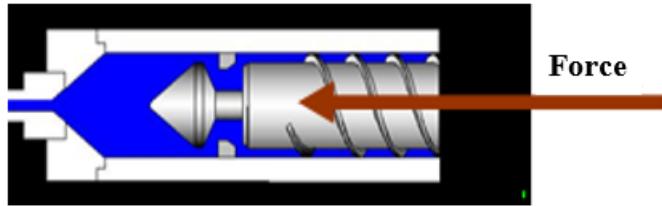
- recovery speed (rpm)
- hydraulic backpressure (psi or bars)
- barrel temperature profile ($^{\circ}$ F or $^{\circ}$ C)
- screw recovery position (in or mm)
- decompression (in or mm)

***Universal* recovery parameters:**

- recovery time (seconds)
- melt backpressure (psi or bars)
- melt temperature ($^{\circ}$ F or $^{\circ}$ C)
- recovery volume (in^3 o mm^3)
- decompress volume (in^3 o mm^3)

Backpressure

Backpressure's objective is to create an opposing force against the free displacement of the screw during recovery. This force will show in the form of melt pressure.



II-3. Backpressure

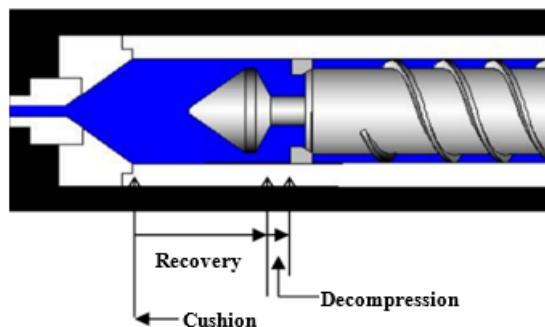
An increase in backpressure results in an increase in friction and an increase in the mixing capacity.

An increase in recovery speed results in an increase in friction and a reduction in the recovery time.

Thermoplastic melts are compressible and some of these materials can be compressed over 20%. Consequently, an increase in backpressure results in an increase in the amount of injected material since more plastic will be compacted into the same volume inside the injection unit.

Decompression after recovery

Decompression reduces the melt pressure after recovery. This is done with a slight screw displacement in the direction of the screw's recovery.



II-4. Decompression

During recovery, the mold is closed and full of material; it is this material that keeps the melt inside the injection unit. After recovery, the melt is compressed; if the mold opens and demolds the finished parts, the melt

could drool inside the open mold. For this reason, there must be decompression after recovery. The only exception would be when the mold includes valve gates, or the injection unit includes a shutoff nozzle; in this case, the decompression after recovery could be set to zero.

There can also be decompression before recovery. Although it is not so common, since the injection barrel is practically empty, this movement helps with sprue removal and makes it easier to close the valve gates in hot runner molds.

Remember:

- If the mold opens during recovery, and neither the injection unit nor the mold include some type of valve system for closing, the melt will drool inside the mold.
- As a general rule, recovery should end about one second before cooling does.
- It is important to know that the injection unit uses two heat sources in order to melt the plastic: heater bands and friction.
- Although the operator programs the heat zones of the barrel, the melt temperature is the most significant factor.

Press Movements

The movements of the press are simple; however, you must be careful during their adjustments or expensive breakage could occur. Each machine has unique programming, and you are responsible for learning and understanding your control's operation before using it.

Let's look at a basic sequence:

- The mold opens.
- The cores, if they exist, open.
- The molded parts are ejected.
- The cores, if they exist, go back into place.
- The mold starts closing.
- The mold protection stage begins.
- The clamping force is completed.

Opening the mold

Permission to open the press occurs after the cooling stage ends and, in most molds, after recovery has ended. The most significant parameters are speed and opening position.

Removing the cores

Not all molds need this option. Cores are mostly used in molds in which the cavity must open or disarm for the parts to be ejected. This type of disarming is operated by an external mechanism, be it hydraulic or electromechanical. This operation can occur as the mold opens.

Ejecting the parts

Normally the molded parts and their runners are held by the mold, and their removal depends upon some type of ejection mechanism. The finished parts are ejected by means of ejector pins, ejector plates, etc. This operation can occur while the mold is opening (“ejecting-on-the-fly”) or after the mold is completely opened. It is common to see robots remove the molded components. The most significant parameters are ejection speed and displacement.

Relocating the Cores

Any molds containing cores must have them return to their position before the mold can close. It is important to connect all signals that guarantee that the core assembly is completed. Mold closure with the cores out of place could cause major damage to the mold.

Closing the Mold

Once the cores and the ejection plates are in place, the mold begins to close. The most significant parameters at this stage are the speed profile and the positions.

Mold Protect

Protecting the mold is extremely important and must be set in such a way as to detect any clamping difficulties which may be due to some trapped object, so that the mold does not get damaged. Some of the most significant parameters are the protection zone, time limit and force limit. For example, during the protection zone, the press will stop closing if the force limit or the time limit is reached.

Reaching the Clamping Force

Once the mold protection stage has passed, the press will continue to close until reaching the adjusted clamping force. Clamping force is adjusted according to the mold's requirements.

You can never be too careful; understand your mold and the machine controls before trying to operate the press. Follow all safety rules provided by the equipment manufacturer and established by your company.

Questions

- 1) Some ***Universal*** parameters are
 - a. injection speed, screw position, hydraulic injection pressure and revolutions per minute.
 - b. injection time, melt volume, plastic injection pressure and recovery time.
- 2) Select the correct sentence:
 - a. Machine parameters are those values that the melt sees before entering the mold.
 - b. ***Universal*** parameters are those values that the melt sees before entering the mold.
- 3) The ***Universal*** parameters of the Injection stage are
 - a. injection pressure.
 - b. injection speed.
 - c. injection time and flow.
- 4) Select the correct sentences:
 - a. Injection speed is a ***Universal*** parameter.
 - b. Flow is the division between injection volume and injection time.
 - c. During the injection stage we want the mold to fill close to 95% of its total volume.
 - d. The units for injection speed are cubic millimeters/second.
- 5) How many speeds are recommended during the injection stage?
 - a. Only one, whenever the transfer position is properly adjusted.
 - b. Two, with a slow ending.
 - c. Three: start slow, continue quickly, and finish slow.
- 6) Do we control injection pressure and time during the injection stage?
 - a. Yes, we control them in order to guarantee the injection speed.
 - b. No, we limit them in order to protect the mold and the machine.
- 7) Select the correct statements:
 - a. Transfer by time measures displacement from the beginning of injection until reaching the entered transfer position.

- b. Transfer by position measures displacement from the beginning of injection until reaching the entered transfer position.
- c. Transfer by pressure monitors the time from the beginning of injection until the entered time expires.
- d. **Universal** molders prefer transferring by position.

8) Select the correct statements:

- a. The transfer position ends the injection stage and starts the hold stage.
- b. Transfer by position is a **Universal** parameter.
- c. The units of transfer by volume are inches or millimeters.
- d. **Universal** molders prefer transfer by position, and if the machine is capable to transfer by volume, then transfer by volume.

9) Select the correct statements:

- a. The hold stage is known as the pressure control stage.
- b. The mold should be filled 100% during the injection stage.
- c. The hold stage is known as the speed control stage.
- d. We should always use multiple hold pressures.

10) Can the cushion equal zero?

- a. Yes, that way you will have control of the mass dimensions.
- b. No, the cushion should always exist so that the hold can work.

11) Select the correct statements:

- a. In the cooling stage, we can correct mass dimensions.
- b. The temperature of the water entering the mold and the temperature of the mold's metal are equal.
- c. The control parameters for the cooling stage are the mold temperature and the cooling time.
- d. The mold temperature and the cooling time are both **Universal** and machine parameters.

12) Select the correct statements:

- a. In the recovery stage we can correct mass dimensions.
- b. The barrel heat zones measure the temperatures of the metal and not of the melt.
- c. Melt temperature is a **Universal** parameter.
- d. Melt temperature is more significant than the barrel heat zones.

- 13) Recovery's *Universal* parameters are
- recovery time, plastic backpressure, melt temperature, and recovery volume.
 - recovery speed, hydraulic backpressure, temperature profile, and recovery position.
- 14) An increase in backpressure will result in
- an increase in friction and a reduction in recovery time.
 - an increase in friction and an increase in the mixing capacity.
- 15) Decompression after recovery
- reduces melt pressure before recovery.
 - reduces melt pressure after recovery.
 - seals the check ring against the screw, keeping melt from passing to the front of the screw.
- 16) An increase in backpressure means an increase in
- the amount of melt during recovery, friction, and velocity (rpm).
 - the amount of melt during recovery, friction, and mixing capacity.
- 17) Thermoplastic melt is compressible.
- True, because melt quantity increases with backpressure.
 - False, it is not compressible, and melt quantity decreases with backpressure.
- 18) Select the correct statements:
- Decompression increases the amount of melt.
 - Decompression at the end recovery reduces melt pressure after recovery.
 - There can also be backpressure before recovery.
 - During recovery, the mold is closed and full of material and it is this material which keeps the melt inside the injection unit.

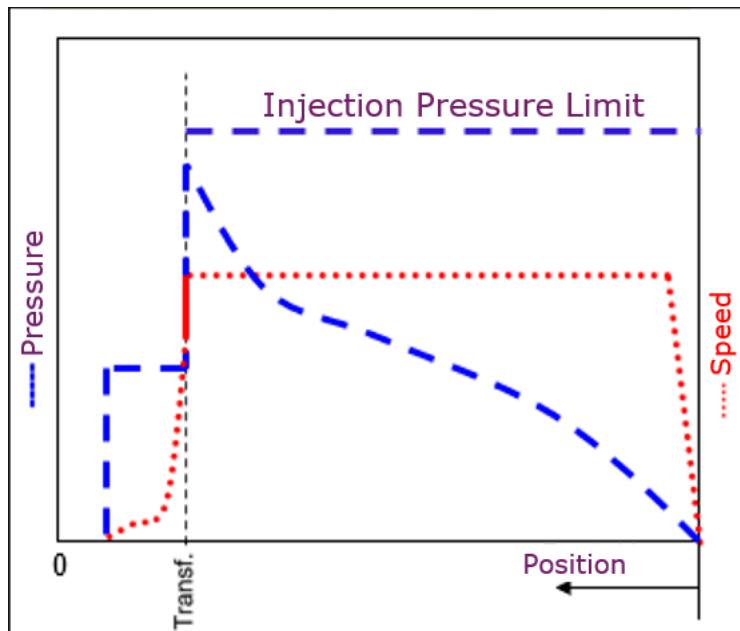
III. Process Graphs

- Molding with Graphs
- PVT Diagrams

Molding with Graphs

If your machine is equipped with injection graphs, you should learn how to use them. Even though the graphs will vary depending upon the equipment manufacturer, they should provide the same information.

Injection graphs describe the behavior of the melt during the time that the mold is filling.



III-1. Graph of ideal injection

The horizontal axis describes the displacement, or position, of the injection screw. The maximum position represents the recovery position plus decompression. The transfer position is the point at which injection ends, and hold begins. The minimum position “0” is the point at which the screw is in its most forward position, with zero cushion.

The right vertical axis represents the speed, or velocity, while the left vertical axis represents injection pressure.

Once injection starts, the screw accelerates from its maximum position until it reaches the adjusted injection velocity; it then maintains that velocity until the transfer position, which is the end of the injection stage.

After transfer, you will observe minimum velocity. Remember that, after the transfer position is reached, the remaining 5% of the mold will be filled and consequently, a minimum screw velocity will be observed.

Note that during the injection stage the pressure continually increases. During the fill, the melt seeks to attach to static surfaces, which could be the mold's walls or on top of plastic already attached to the mold's surface, and it will resist flowing. Since pressure is the result of this flow resistance, the pressure will increase as the amount of material in the mold increases.

If the pressure decreases, it may be due a decrease in the injection speed (flow). Previously we established that **Universal** molders use one single velocity, with very few exceptions. Under normal conditions, pressure will always increase until reaching the transfer position, the position where the mold is filled close to 95%. Do not try to overfill the mold during the injection stage; a 95% fill is enough. Do not try to maximize this amount; there will be no real benefit.

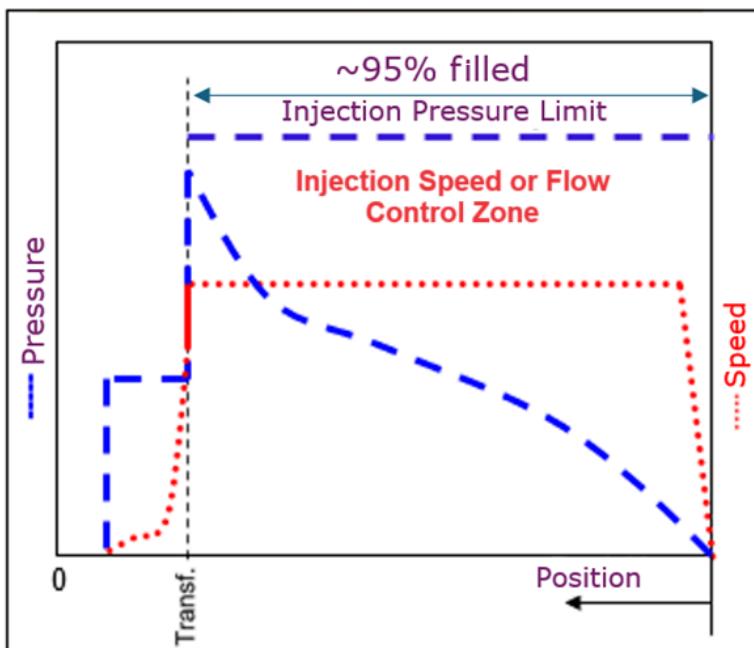
Some molds exhibit extreme difficulty in filling: for example, nylon ties which are long and thin, or micro-molding applications with narrow and awkward spaces for filling. In such cases, a filling percentage greater than 95% may be required.

Also, remember that the injection pressure limit should always be greater than the injection pressure. Program the injection pressure limit to be 5% to 10% higher than the injection stage pressure.

It is important to know that some controllers are equipped with more than one injection pressure limit, one for each injection velocity. Remember that **Universal** molders try to mold with only one injection velocity. Now, if your controller requires more than one pressure limit, program them to be 5 to 10% above the maximum injection pressure.

For example, if the injection pressure reaches the injection pressure limit because the cavity gate is blocked with solidified material, the screw will

still try to reach the transfer position. If the limit is not properly adjusted, the pressure will continue to increase and the excess melt, corresponding to the blocked cavity, could surpass the clamping force and cause the



mold to open.

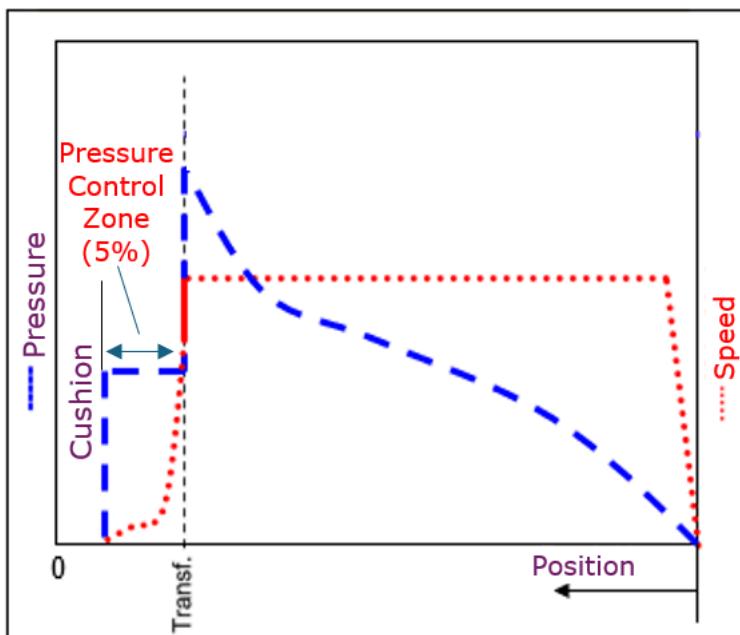
III-2. Control zone graph of injection speed or flow

In summary:

- During the injection stage, approximately 95% of the volume of the mold (cavities and runner) is filled.
- During the injection stage, the injection pressure limit is programmed, which should never be reached since its only purpose is to protect the machine and the mold.
- Pressure is the result of the opposition to flow; the more the melt enters the mold, the more the opposition increases. Assuming that there is only one injection speed, the pressure will increase until reaching the transfer position.
- The injection pressure limit should be programmed to 5 to 10% more than the injection stage pressure.
- The cushion position should be less than the transfer position.

In some processes with the holding stage off (where the holding pressure is set to zero) or with a very low hold pressure (5% of the pressure at the time of transfer), it's possible that the cushion position might be greater than the transfer position. This phenomenon has been observed in servo-electric injection units, where the servo control tries to reduce the holding pressure (programmed to zero or very low) by retracting the screw. As a result, the cushion position could end up above the transfer position, even if the injection was adjusted to 95% of the total fill. Furthermore, this effect has also been observed in hydraulic injection units. In this case, the holding pressure is significantly lower than the pressure at the time of transfer, which pushes the screw to a position higher than the transfer position. This phenomenon is intensified in hydraulic units when attempting to fill beyond 95% of the fill.

After transfer, the hold stage/zone is started. In this stage, we control pressure, and velocity is a result. The cavities are packed and held until the gates solidify.



III-3. Graph of hold zone or pressure control

We can see that the pressure is being controlled and the velocity, although minimum, shows some movement.

It is not normal for the hold pressure to be greater than the pressure at transfer. We have found processes in which the hold pressure was greater and have discovered that they were filling much less than 95% of the melt required during the injection stage. Once the transfer and recovery positions were properly adjusted, the hold pressure was less than the transfer pressure. How much less should the hold pressure be than the transfer pressure? It will depend on the mass dimensions.

These graphs should be similar when the molds include hot runners. The difference between cold and hot runners is the objective of reducing material consumption since the hot runner never solidifies. The melted material in the hot runners will be part of the next cavity fill; even with hot runners you still must inject, pack and hold until the gates freeze.

There exist some molds with hot runners that include valves in each gate. When activated, these valves create the gate freeze effect. Their operation is simple, once the cavities are packed, the valves in each gate are activated, sealing them so no melt can pass. The signal that closes the valves originates from the signal that corresponds to the hold time. Once these gate valves have completed their function they open, permitting fill like any other gate.

The objective of these valves is one of providing an improved cosmetic finish of the molded part fill point and a reduction of the hold time since there is no need to wait for the gates to solidify.

In summary:

- During the hold stage, velocity is the result of pressure control, and its magnitude will be insignificant.
- During the hold stage the remaining cavity volume, which had not been filled during the injection stage, will be filled.
- The hold pressure is maintained until the gates solidify.
- In this stage we only consider filling the cavities; the effect that we are looking for is the mass dimensions of the parts in the cavities. Remember that we are molding parts and not runners.
- The farthest position, which should never be equal to zero, is called the cushion. If you reach the zero position, that could mean that the

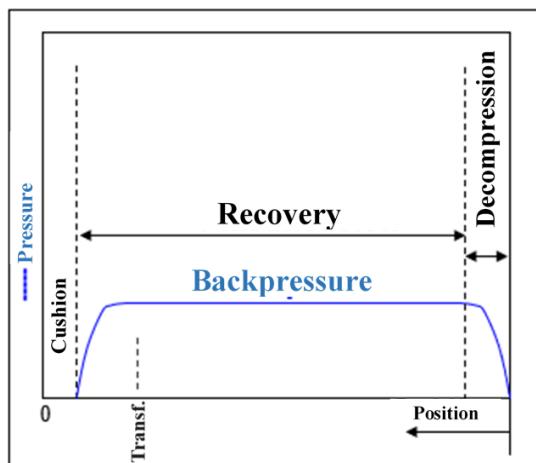
process is out of control. One reason could be a defective check ring, possibly scratched or worn, that is not properly sealing against the screw.

Identify each control zone:

- Injection: the zone that controls injection velocity (flow).
- Hold: the zone that controls hold pressure and time.
- Cooling: the zone that controls the temperature of the mold and cooling time.

Recovery and decompression occur during the cooling stage.

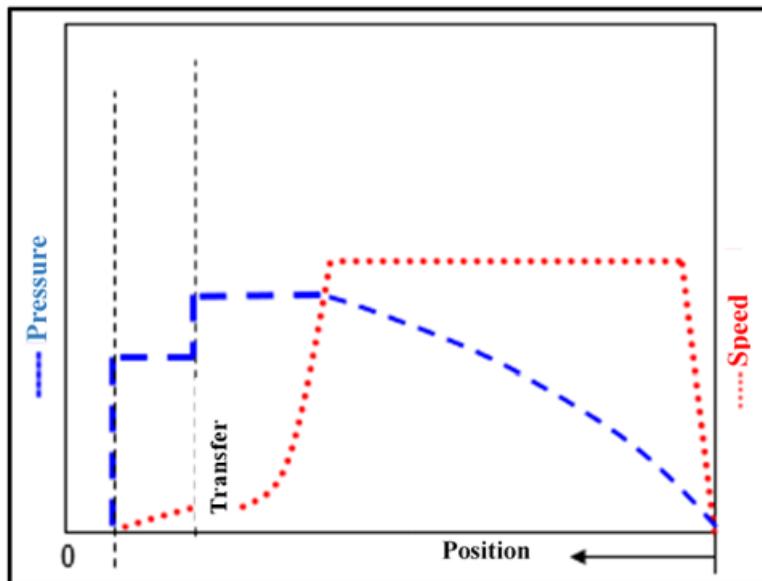
Recovery happens when the screw rotates and fills melt in the barrel for the next cycle. The graph below illustrates the pressure during recovery, which should be equal to the adjusted backpressure. Once the screw fills with the required volume, the plasticized material is decompressed in order to avoid drooling during demolding.



III-4. Graph of recovery zone

One of the advantages that we obtain from molding by graphs is that with a simple glance we can determine if the process is within or out of control, and even people that are not necessarily molders could interpret the process's behavior in each stage of the graph.

The following graph illustrates the results of an incorrectly programmed, limited injection pressure.

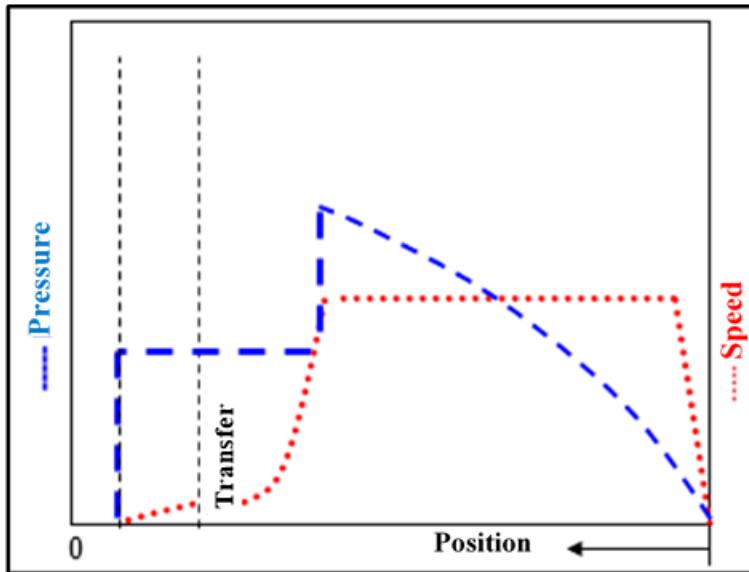


III-5. Graph with limited pressure

You can clearly observe that the injection speed is out of control due to the injection pressure limit being adjusted too low. The injection pressure limit should be adjusted to 5 to 10 % higher than the injection pressure required to fill the mold, in order to control the speed.

Limiting the injection pressure is a practice used for defective molds that create flash. If your industry allows you to operate molds under this situation, and you find yourself having to operate with this condition, consider reducing the injection speed. Although this is not the best scenario, you will be controlling the speed. Understand that the injection pressure is reduced when you reduce the injection speed.

The following graph illustrates the results of a premature transfer.

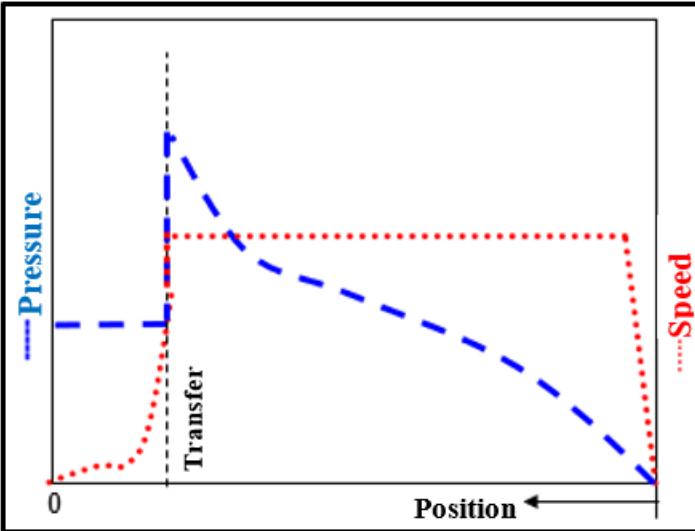


III-6. Graph illustrating premature transfer

For some reason, the transfer occurred prematurely; this could have been the result of a second transfer method that was programmed. For example, if transfer by position and transfer by pressure were programmed, the first of these conditions to be met would initiate the transfer. If this were the case, the transfer would occur based on pressure and not position.

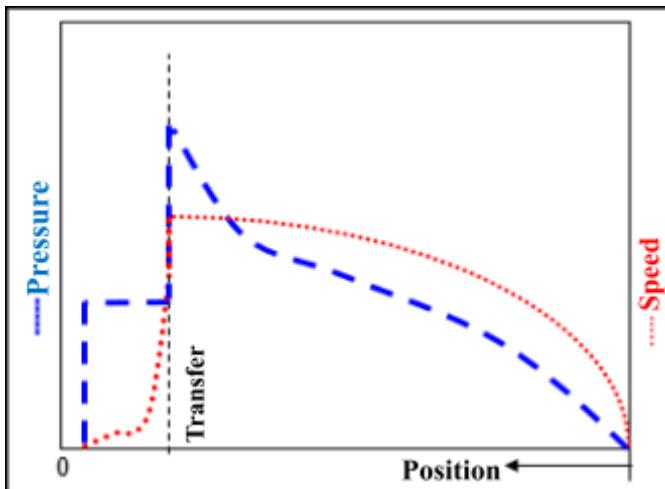
A graph showing a defective process due to zero cushion is illustrated in the following figure. It clearly shows that the screw moved completely forward, emptying the barrel entirely. No cushion exists, and there is no control of the mass dimensions. This could be caused by a worn check ring.

It is important to remember that the cushion position should always be less than the transfer position and should never be equal to zero.



III-7. Graph illustrating cushion equal to zero

The following graph illustrates a defective process due to a limited injection speed. Here the operator was not aware that the programmed injection speed was not being reached or did not verify the fill graphs. This process is clearly out of control and the injection time, a highly significant value, could fluctuate uncontrollably. As a general rule, you do not program process values that cannot be reached.

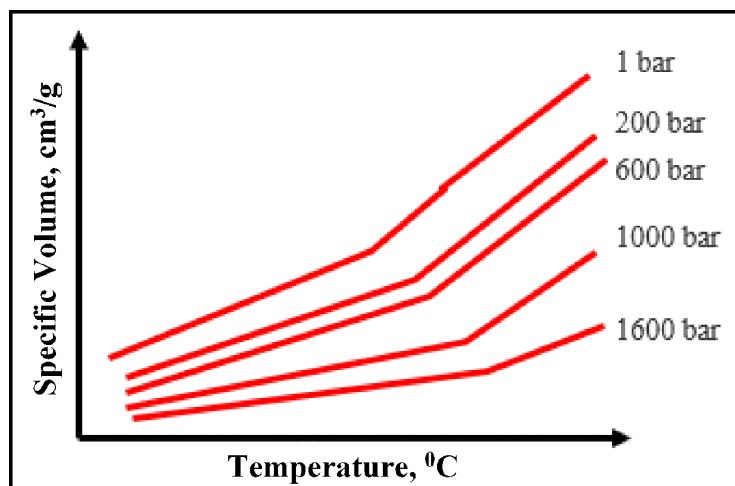


III-8. Graph illustrating unachieved programmed injection speed

Molding with graphs is a powerful tool. Understand your equipment and strive to maximize the utilization of the graphs. Copy and enlarge these graphs and post them throughout the molding room, and you will appreciate their benefits.

PVT Diagrams

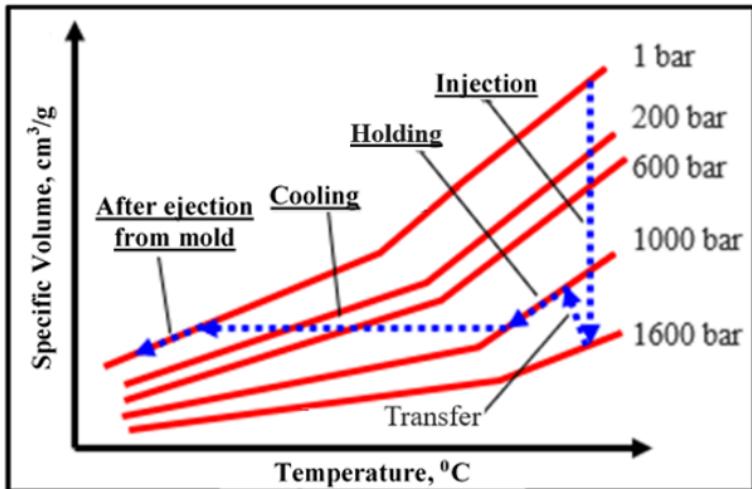
For the benefit of the molders that use pressure and temperature sensors in the cavities, we will take these concepts to a PVT (Pressure, Specific Volume and Temperature) diagram.



III-9. PVT (Pressure, Specific Volume and Temperature) diagram

The vertical axis represents the specific volume, the inverse of density: specific volume = volume/mass. The horizontal axis represents the melt temperature. The solid lines represent grafts of constant pressure, from 1 bar to 1600 bars. Note that the value of pressure increases as you go down to the next line. The movement within a graph line represents a specific volume at a unique melt temperature.

Note in the following PVT diagram the dashed lines representing the molding stages.



III-10. PVT diagram with molding stages

Before injection, the material decompresses. Once the injection stage begins, the pressure increases, compacting the material at a relatively constant temperature.

Once the transfer position is reached the pressure control zone begins, normally at a pressure lower than the transfer pressure.

The cavities are packed at a constant pressure; consequently there will be more material per unit of volume, causing a reduction in specific volume. During the holding stage the melt loses heat, and as is expected, the graph illustrates a reduction in temperature.

Once the gates have solidified, the holding stage ends, and the cooling stage begins. The parts remain pressurized and restricted inside the cavities while cooling. The heat is rapidly removed with a minimum reduction in specific volume. The compressed melt solidifies, and the pressure decreases.

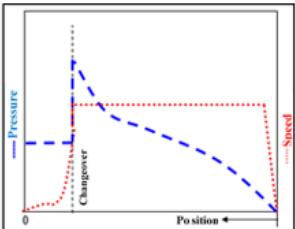
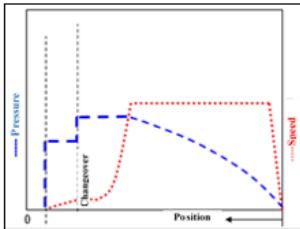
Once the cooling time expires, the parts are ejected from the mold into the room's environment where they continue to cool and shrink at a constant pressure.

Once again, molding with graphs is a powerful tool; utilize it. It is a fast and effective method of verifying the processes even when you are not a molder.

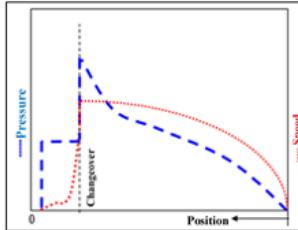
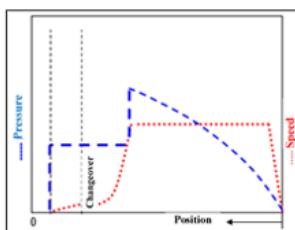
Questions

- 1) During the injection stage we control injection pressure.
 - a. True, we control pressure and temperature.
 - b. False, we control injection velocity.
- 2) We can fill 100% of the cavities during the injection stage.
 - a. True, 100% is completed in the transfer.
 - b. False, 100% is completed during the holding stage.
- 3) The injection stage should end when
 - a. the injection time expires.
 - b. the pressure limit is reached.
 - c. the transfer position is reached.
- 4) In the holding stage we achieve
 - a. thermal dimensions.
 - b. mass dimensions.
 - c. constant speed.
 - d. a homogeneous melt.
- 5) In the holding stage we control
 - a. backpressure.
 - b. fill velocity.
 - c. hold pressure and hold time.
- 6) The holding stage should end
 - a. when the gates solidify.
 - b. by position.
 - c. by pressure.
- 7) In the cooling stage we control
 - a. fill time and the pressure limit.
 - b. the flow of oil in the hydraulic cylinder.
 - c. cooling time and the mold temperature.
- 8) In the cooling stage we want
 - a. to eject parts with adequate mass dimensions.
 - b. the external dimensions.

- c. to eject parts with adequate thermal dimensions.
- 9) The cushion can equal zero.
- a. True, that way we control mass dimensions.
 - b. False, we cannot control mass dimensions.
- 10) Pair the graphs of out-of-control processes with their respective flaw description.



- a. Zero cushion
- b. Programmed velocity was not reached
- c. Premature transfer
- d. Limited pressure



IV. Plastic Morphology

- **Types of Plastics**
- **Mechanical Properties**
- **Common Materials and their Characteristics**
- **Some Experiments**

Types of Plastics

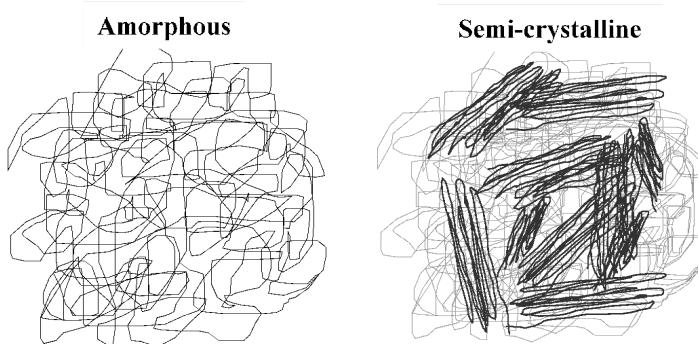
Basically, there exists two types of plastics: thermosets and thermoplastics.

Thermoset materials are those materials that can be molded only once. The injection molding of thermosets is somewhat different than the molding of thermoplastics; therefore, **Universal Molding™** for thermosets such as liquid silicon, bulk molding compounds (BMC), epoxy, rubber, melamine, etc., will not be discussed in this text.

Thermoplastic polymers are those materials that can be melted repeatedly, and consequently can be reground and molded multiple times. Their molecular organization falls into two types, amorphous and semi-crystalline.

Amorphous materials are thermoplastics that do not offer any type of molecular organization; in other words, their molecules have a totally random organization, like spaghetti.

Semi-crystalline materials are a combination of organized segments (crystals) surrounded by disorganized (amorphous) segments.



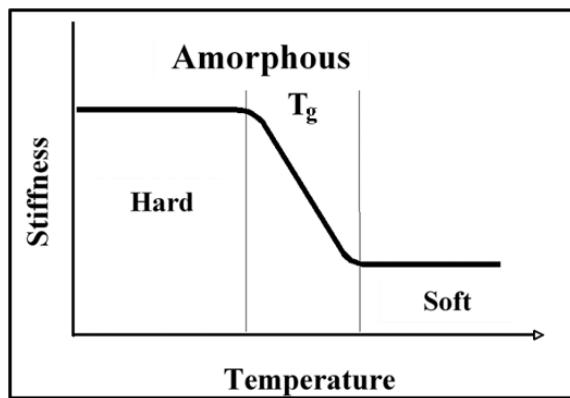
IV-1. Illustration representing amorphous and semi-crystalline molecular organization

The molecular structure of semi-crystalline material includes organized groups with some type of orientation, known as crystals. These crystals are surrounded by disorganized amorphous molecules. This partially

organized mix of amorphous and crystalline material is known as semi-crystalline.

Mechanical Properties

A material's molecular orientation brings mechanical properties which can affect the injection molding process. The mechanical characteristic of these materials can be illustrated with a graph of stiffness versus temperature. Look at the graph below.

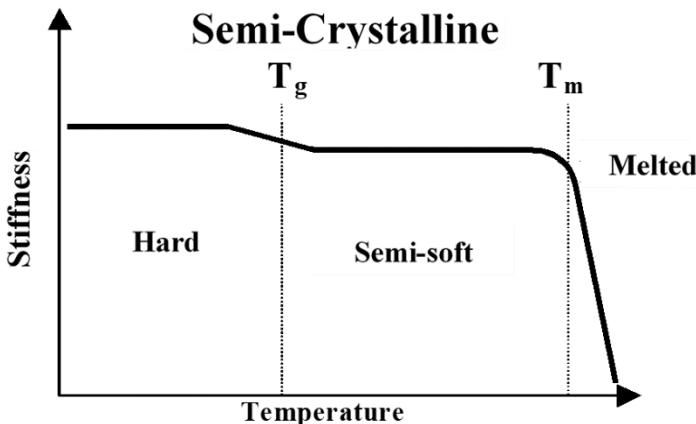


IV 2. Graph illustrating stiffness versus temperature in amorphous materials

The vertical axis indicates the stiffness, in which the origin represents soft or melted material and any increase represents a rise in stiffness. The horizontal axis indicates the material's temperature, in which the origin represents a material at room temperature and any increase represents a rise in temperature.

The profile graph of amorphous polymers illustrates that, at certain low temperatures, the material will stay in a solid state. With an increase in temperature the material will reach a glass transition temperature T_g . After this glass transition temperature is reached, the material enters a transition zone that is known as the glass transition zone, where it will gradually lose its stiffness. If the temperature continues to increase, you will obtain a totally soft material. Think of it as an elastic or putty-like melt that is not liquid. It is in this softened state that amorphous materials are injected.

The graph of a semi-crystalline material offers a distinct picture.



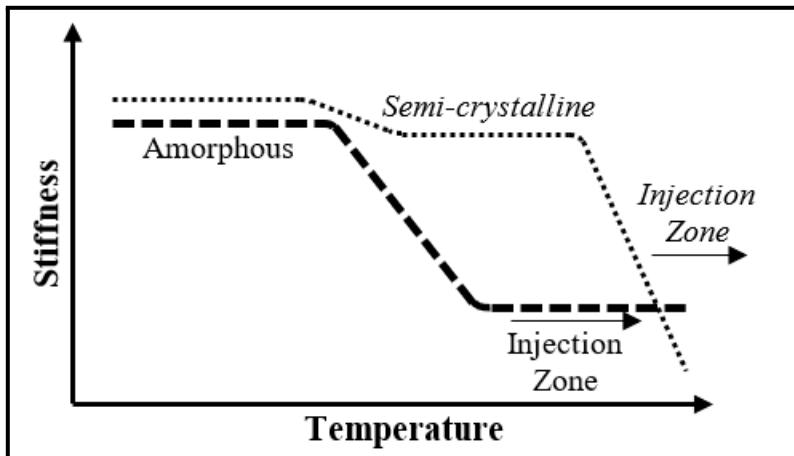
IV-3. Graph illustrating stiffness versus temperature in semi-crystalline materials

The graph profile of a semi-crystalline material shows that, just like amorphous polymers, the material will remain in a solid state at lower temperatures and reaches a glass transition zone at temperature T_g when the temperature is increased. With semi-crystallines, the glass zone corresponds to the amorphous part of the material and is so insignificant that the term T_g is rarely used.

If the temperature continues to increase beyond the glass transition zone, the material will lose some stiffness. Even then, it will remain in a solid state.

If the temperature keeps increasing, melt temperature T_m will be reached. At temperatures greater than T_m semi-crystalline materials will melt. After T_m the material becomes liquid; for this reason, it is said that semi-crystallines melt and do not soften like amorphous materials. It is after temperature T_m that semi-crystallines are molded.

Note that semi-crystalline materials do not have the advantage of as large an injection zone as amorphous materials, making them harder to melt. Semi-crystalline materials change from melted to solid more quickly than amorphous materials do.



IV-4. Overlaid graphs of stiffness versus temperature of semi-crystalline and amorphous materials

Once the material has been identified, determine the injection molding process settings, taking into consideration that amorphous materials soften and semi-crystalline materials melt

Common Materials and their Characteristics

Amorphous	Semi-crystalline
ABS	PA (nylon)
Polystyrene	Acetate
Acrylic	Polyethylene
PVC	Polyester
Polycarbonate	Polypropylene

IV-5. Common amorphous and semi-crystalline materials

Semi-crystallines are only opaque in their solid state, since in their melted state the crystals become disorganized like an amorphous material and permit the passage of light. If you ever have the opportunity to observe a purge of a semi-crystalline melt, for example, polyethylene with no

additives, you can appreciate how the material changes from translucent to opaque as it cools and solidifies.

Not all melted materials have amorphous morphology; there are some that form crystals in their liquid state. These materials are called *liquid crystal polymers*.

The formation of crystals in semi-crystalline materials can be controlled by adjusting the molding process. In other words, as a molder, you can control the quantity of crystals formed during the cooling stage. Being able to control the formation of crystals with adjustments in the molding process is an opportunity. For example:

- With the objective of maximizing clarity in the fabrication of PET preforms, which will later be stretched and blown to form bottles, crystal formation needs to be reduced as much as possible. If you ever look at PET pellets you will notice that they are white and opaque, the PET having been crystallized to its maximum. Remember that crystals block the passage of light. That is why during cooling a thermal shock is given, using a super cold mold, with the objective of halting the formation of crystals.
- The ultrasonic assembly of thermoplastics can be damped by the material's crystalline structure. This is another reason some molders control the formation of crystals.
- When it is necessary to control mechanical properties such as flexibility and toughness in molded products made from semi-crystalline resins, such as nylon ties for cable bundling (Tie wraps), it is important to consider the material's crystalline structure. This crystalline structure tends to make the material more rigid and inflexible. For this reason, some molders implement strategies to control crystal formation during the molding process.

Know the type of material you are using before trying to mold a product and identify the mechanical properties that most affect your product.

Remember that hold pressure and cooling temperature combine to affect the final dimensions of a molded product. The cooling temperature affects the thermal dimensions, and the hold pressure affects the mass dimensions. In addition, the temperature and pressure could create a

combined effect; this is when the temperature and the pressure combine within a certain range, creating a particular effect on the dimensions. Although the combined contribution is difficult to visualize, it does exist. For example, the time it takes to solidify the gates corresponds the hold stage and could be affected by the mold temperature, a cooling stage parameter.

Amorphous	Semi-crystallines
<u><i>Soften</i></u> When you inject an amorphous melt, it has a putty or gummy consistency. A purge of this type of melt will clump like a paste with little intention to flow.	<u><i>Melt</i></u> When you inject a semi-crystalline melt, it has a liquid consistency. In other words, a purge of this type of melt will run like a liquid.
<u><i>Less resistant to chemical attacks</i></u> When amorphous materials are exposed to a chemical like a solvent, they could decompose. Even though polycarbonate is used for bulletproof glass, a polycarbonate container will decompose if filled with gasoline.	<u><i>More resistant to chemical attacks</i></u> Semi-crystallines have a better resistance to solvents. For example, an HDPE container can store gasoline.
<u><i>Transparent</i></u> Melted or solid amorphous materials without additives are translucent or clear. Note: Do not confuse the “material clarity” property with the term “crystalline morphology”.	<u><i>Opaque</i></u> Semi-crystallines in their solid state have organized segments (crystals), which divert the passage of light and thus are opaque.
<u><i>Low shrinkage</i></u> Amorphous materials shrink less since the disorganized molecules occupy more space. Note: It is said that mass dimensions are more significant with amorphous materials.	<u><i>High shrinkage</i></u> The crystals in semi-crystallines are groups of organized molecules that tend to take up less space. This is why they shrink more during cooling and crystal formation. Note: Thermal dimensions are said to be more significant with semi-crystalline materials.

IV-6. List of mechanical characteristics of amorphous and semi-crystalline materials

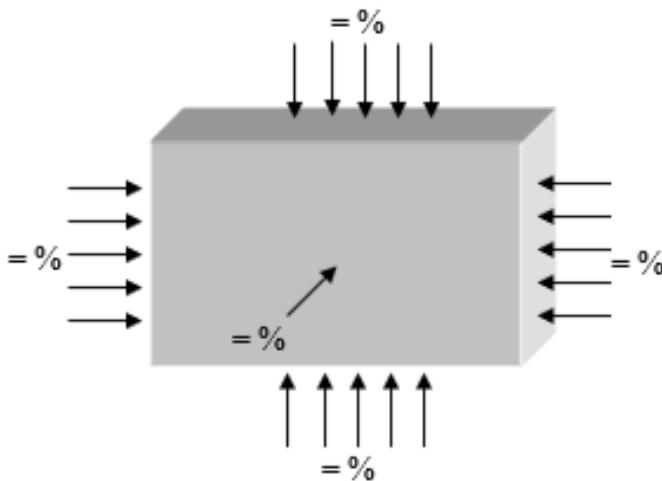
The contribution to the dimension of each of these characteristics will depend upon the type of material being used. Because of this you should understand your material before trying to correct dimensions or any attribute.

Amorphous	Semi-crystalline
Problems of overpacking due to low shrinkage.	Problems with incomplete packing due to too much shrinkage.
Problems with flash due to overpacking.	Parts with sink marks due to high shrinkage.
Parts breaking during ejection since little shrinkage causes parts to stick to the cavity walls.	Easy part ejection since high shrinkage helps separate parts from the cavity walls.

IV-7. List of process characteristics of amorphous and semi-crystalline materials

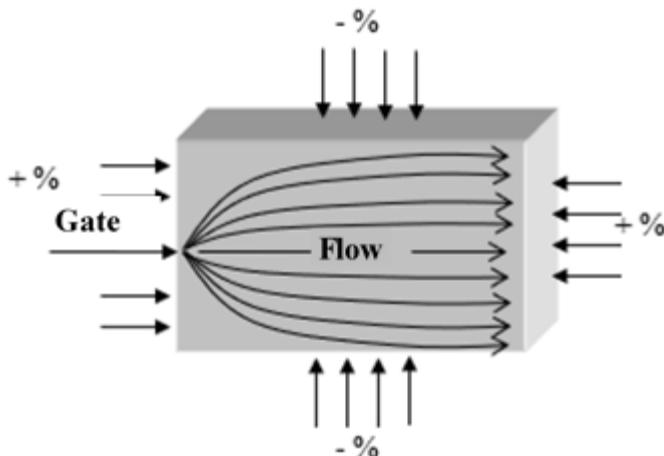
Shrinkage

Amorphous materials, for practical purposes, shrink proportionally in all directions.



IV-8. Amorphous material shrinkage

Semi-crystalline materials shrink more in the direction of injection flow rather than in the direction perpendicular to the injection flow.



IV-9. Semi-crystalline material shrinkage

During injection, the polymeric chains orient themselves to the direction of flow and are forced to maintain some orientation, crystals included, as they solidify. During shrinkage, these molecules seek to conform into a less uncomfortable position; because of this, the shrinkage will be greater in the direction of the flow.

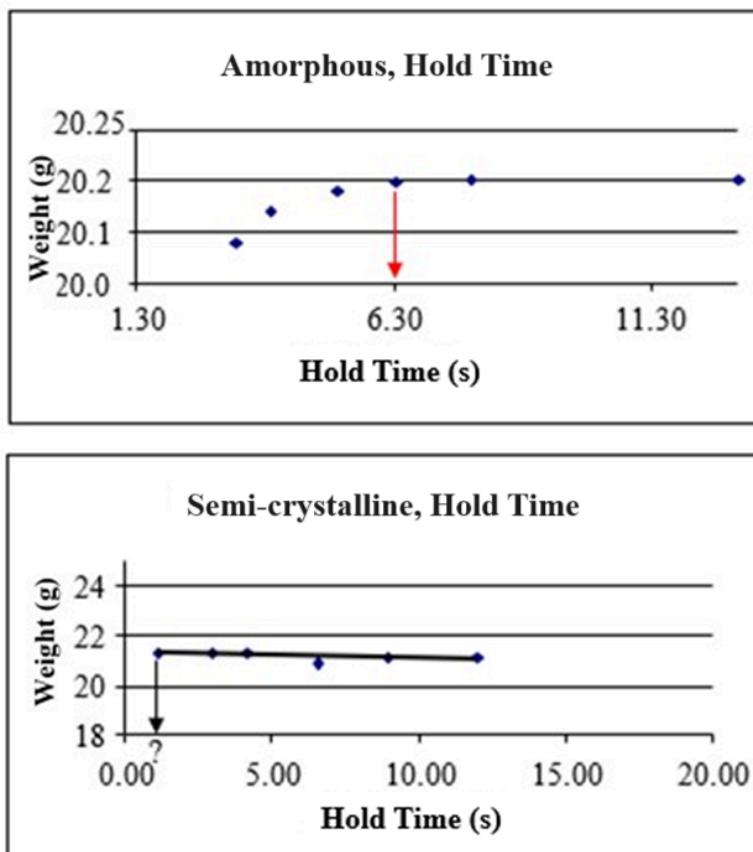
When a semi-crystalline is reinforced with fiberglass its shrinkage becomes inverse, less in the direction of injection flow and greater in the perpendicular flow direction. This is because the glass fibers orient themselves to the direction of injection flow and, during shrinkage, they act like steel bars inside concrete, resisting shrinkage in the direction of their orientation.

Some Experiments

Let us review some experiments performed by *Universal MoldingTM* students.

Experiment #1:

Using the same mold, injection molding machine and auxiliary equipment, the effect of hold time upon gate freeze was tested using two materials, one amorphous (polystyrene) and the other semi-crystalline (nylon).



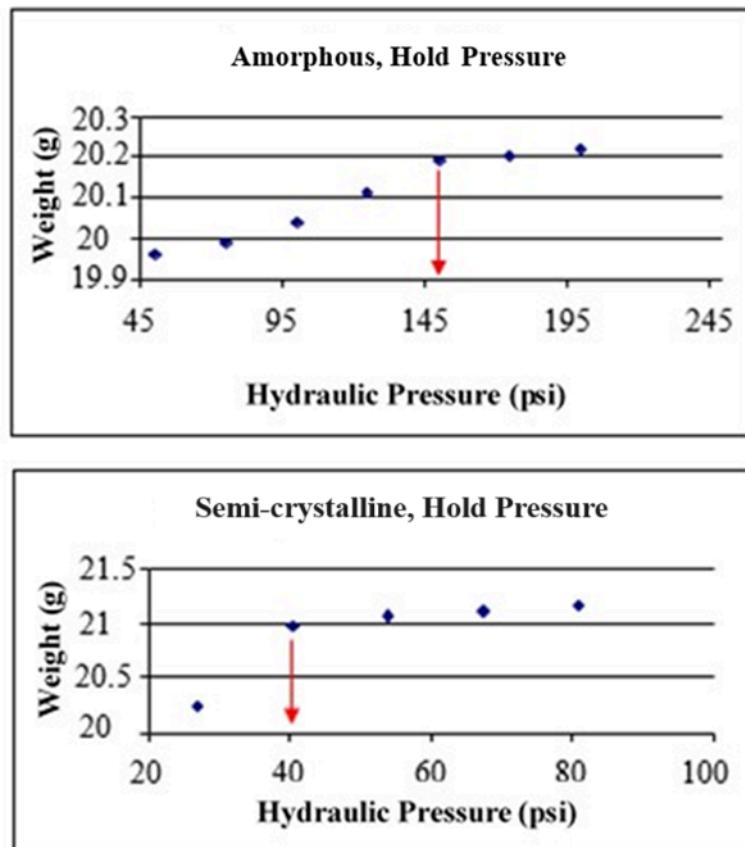
IV-10. The effect of hold time on the part weight in amorphous and semi-crystalline materials

The graphs of part weight versus hold time reveal that the amorphous material solidified in the gates in close to 6 seconds, which consequently caused the weight of the part to stop increasing. With the semi-crystalline material, the students stopped the experiment after one second of packing with the gates solidifying, showing that the gate freeze occurs in less time than with the amorphous material.

This is expected since the semi-crystallines change from solid to liquid and vice versa quickly, unlike amorphous materials that remain in a pasty state during a large temperature range.

Experiment #2:

Using the same mold, injection molding machine and auxiliary equipment, the effect of the hold pressure upon the weight of the parts was tested using two materials, an amorphous (polystyrene) and a semi-crystalline (nylon).



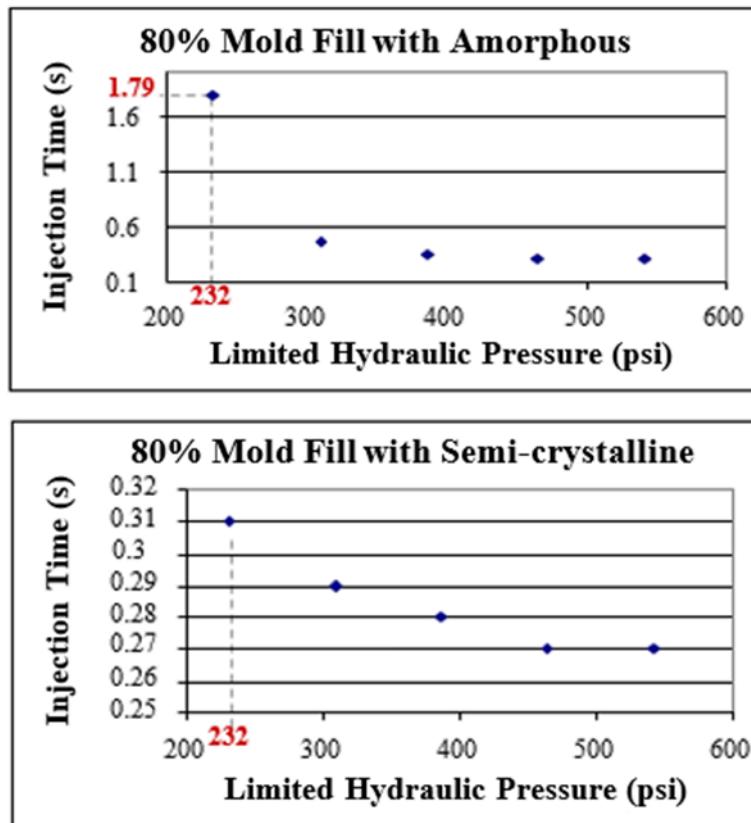
IV-11. The effect of hold pressure on the part weight of amorphous and semi-crystalline materials

The amorphous part's weight stopped increasing significantly at a hold pressure close to 150 psi, while the semi-crystalline part's weight stopped increasing significantly at a hold pressure close to 40 psi.

This holding pressure differential could be attributed to the fact that the amorphous melt is pasty and not liquid like the semi-crystalline, or that a rapid gate freeze occurs when using a semi-crystalline material.

Experiment #3

Using the same mold, injection molding machine and auxiliary equipment, the effect of fill pressure on fill time on a mold filled to 80% was tested with two materials, an amorphous (polystyrene) and a semi-crystalline (nylon).

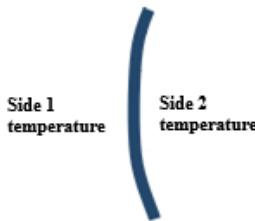


IV-12. The effect of injection pressure on injection time with amorphous and semi-crystalline materials

The semi-crystalline material, being more liquid, required less pressure. For example, at a pressure limit of 232 psi, the fill time with the semi-crystalline was 0.31 seconds compared to 1.79 seconds with the amorphous material.

Questions

- 1) How many times can you melt an already molded thermoset?
 - a. One time, since it was already molded once.
 - b. None, since it was already molded once.
 - c. Four to seven times.
 - d. It depends on the type of thermoset.
- 2) In the recovery stage
 - a. amorphous material melts.
 - b. semi-crystalline material softens.
 - c. semi-crystalline material melts and amorphous material softens.
- 3) To reduce the formation of crystals in semi-crystalline parts that will be ultrasonically welded
 - a. heat the mold to reduce the crystals.
 - b. cool the mold to increase the crystals.
 - c. freeze the molecules as quickly as possible, to minimize the formation of crystals.
- 4) A part warps after being ejected. We know that we can eliminate that warpage by using distinct temperatures on the faces of the mold.



The solution is

- a. make side 1's temperature greater than side 2's temperature.
 - b. make side 2's temperature greater than side 1's temperature.
 - c. make both temperatures equal.
- 5) We control amorphous material shrinkage by
 - a. increasing the mold temperature to get parts with thicker walls.
 - b. heating the mold to halt part shrinkage.
 - c. using hot molds to get more shrinkage and cold molds to get less shrinkage.

- d. none of the above. Amorphous materials do not shrink.
- 6) The shrinkage of semi-crystalline materials is
- a. equal in all directions.
 - b. less in the direction of injection flow.
 - c. greater in the direction of injection flow unless the material is reinforced with fiberglass.
- 7) Since amorphous shrinkage is less than semi-crystalline,
- a. amorphous parts have more sinkage problems.
 - b. semi-crystallines have more problems with part ejection.
 - c. semi-crystalline parts have more sinkage problems than amorphous parts.

V. Auxiliary Equipment

- **Material Drying**
- **Blending and Material Handling**
- **Controlling the Water Temperature to the Mold**

Experience has taught us that a great number of molders place more importance on their molding machines and molds, and not enough on their auxiliary equipment. Before starting to mold, you must be sure that the auxiliary equipment being used in your process is properly sized and adjusted. Understand your equipment and use it properly; selecting auxiliary equipment such as material dryers and material handling equipment is an integral part of the process.

Material Drying

There are certain polymeric materials that need to be dried before being processed. These materials are known as hygroscopic materials and, by the nature of their chemistry, they tend to absorb water from their environment. Water molecules that remain inside these polymers can cause imperfections in molded parts. Some common hygroscopic materials are PA, polycarbonate, PET, ABS, etc.

The material gets moisture from the atmosphere. Molecularly speaking, molecules of hygroscopic materials prefer to be connected to water molecules. During plastic manufacturing at high temperatures and pressures, the material releases these attached water molecules. Once polymerized and brought to normal environmental conditions, the material again pulls water from the atmosphere.

The most common drying imperfections are streaks and bubbles. The streaks, like bubbles, can usually be seen by the naked eye. The streaks are presented in the form of discolored veins in the direction of the flow. Bubbles are the product of vaporized water that ends up being trapped inside the molded parts. One method used to identify these types of bubbles is that of heating the molded parts since the bubbles, as a result of humidity, tend to expand or grow when the parts are heated close to plastic deformation temperatures.

The bubbles weaken the molded parts and become concentration points for force that could cause fractures once the molded parts are exposed to functionality tests.

There are non-hygroscopic materials that also require drying, for example, parts that demand high clarity. Their resin is sometimes dried in

order to maximize translucency. However, this type of moisture is removed just by using heat, since the humidity on the parts is superficial.

Drying mechanics are based on four factors:

- extremely dry air
- drying temperature
- drying flow
- drying time

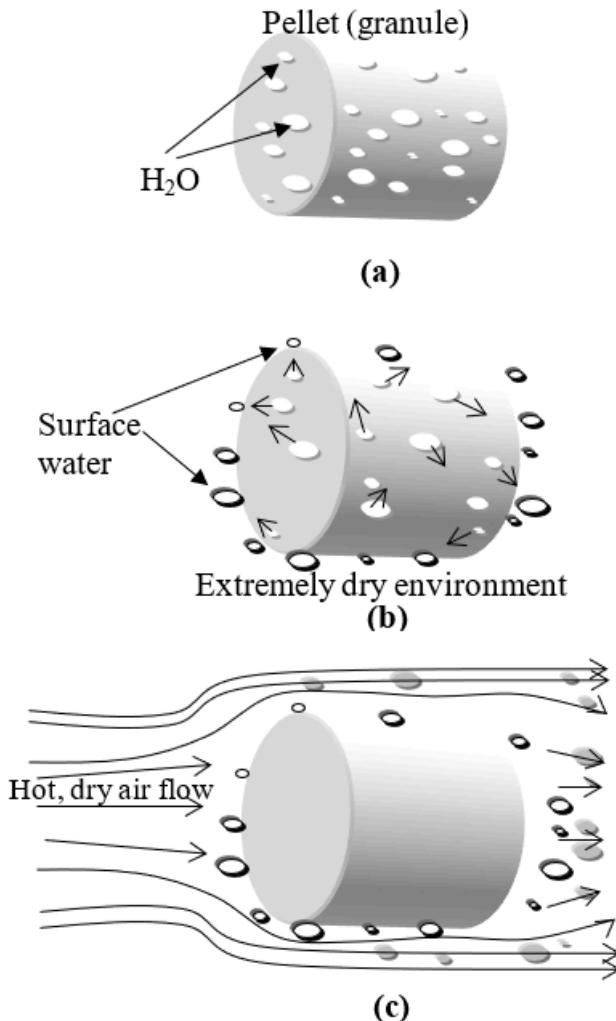
Extremely dry air

An environment with extremely dry air must be created. Moisture is forced to move to the surface of the plastic by means of a super-dry air that is in search of moisture. This air is so dry that no human being could live in it. This is achieved by maintaining a condensation temperature, or dewpoint, less than -30°C (-20°F). Dewpoint is the point at which the water vapor that normally exists in the air condenses. For example, morning dew is a consequence of water condensation caused by a decrease in the nighttime temperature.

Drying temperature

The dryer, besides producing super-dry air, heats that air in order to help extract water from the material's surface and evaporate it. This air temperature is known as the drying temperature and depends on the material being used. For example, polycarbonate requires a drying temperature of around 121°C (250°F).

Drying flow



V-1. The mechanics of drying thermoplastics

- thermoplastic pellets with moisture housed inside the plastic
- An extremely dry environment forces water molecules to come to the surface.
- A flow of hot, dry air evaporates surface water and removes it in the form of water vapor.

There must be enough airflow to remove water vapor from the drying hopper. This super-dry, heated airflow is responsible for transporting water vapor away from the material.

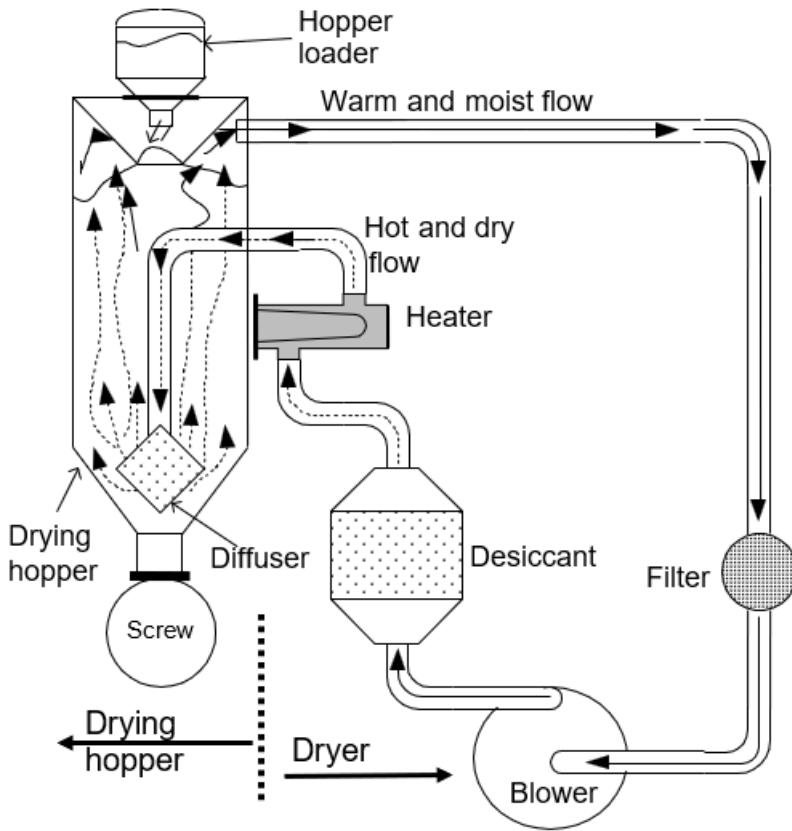
The most common units of flow are cubic meters per minute (cmm) and cubic feet per minute (cfm). This flow depends on material type and material consumption. For example, PA (nylon) requires a dry air flow of 25 liters/min for each 1 kg/hr, or .90 cfm for each 1 lb/hr, of material.

Drying time

The drying process is lengthy and requires enough drying time to complete the removal of moisture. In other words, from the time the material enters the dryer, a defined time must pass so that the resin is exposed to a dry and hot air flow until the drying process is completed. The drying time depends on the material; for example, polycarbonate needs 4 hours of drying. There are many texts that summarize the values of time, flow, and drying temperature; you should still verify them with the data sheet provided by the resin manufacturer.

Drying equipment components

The main components of the drying circuit are the drying hopper and dryer.



V-2. Drying hopper and dryer

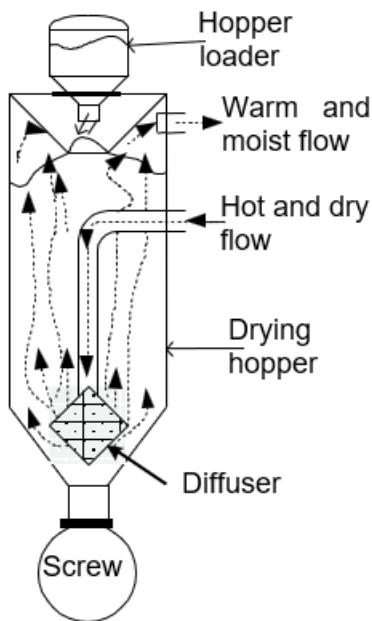
Drying hopper

The drying hopper consists of a resin loader, an inlet for the hot and super-dry air, a diffusor for the hot and super-dry air, an outlet for the returning warm and humid air, and a discharge to the extruder.

Plastic resin resides in the drying hopper while moisture is removed. Hot and super-dry air is forced into the hopper and distributed throughout the plastic resin by a diffuser. This hot, dry air removes moisture and transports it out of the hopper. The humid, warm air flow returns to the dryer.

We can assume a loss of about 56°C (100°F) during drying. This information will be used later.

Once the moisture is removed, the material is ready to be injected.



V-3. Drying hopper

Drying hopper size

The hopper must be properly sized to ensure drying time. For example, a high-consumption process demands a larger hopper than a low-consumption process using the same material.

Drying hoppers are identified by their volume; although some manufacturers use either lb or kg units to identify their hoppers, volumes such as ft³ or liters are more proper.

To determine the size of a dryer hopper, you need to know:

- the material consumption, in lb/hr or kg/hr

$$\text{Material consumption} = (\text{amount of material per cycle}) / (\text{cycle time})$$

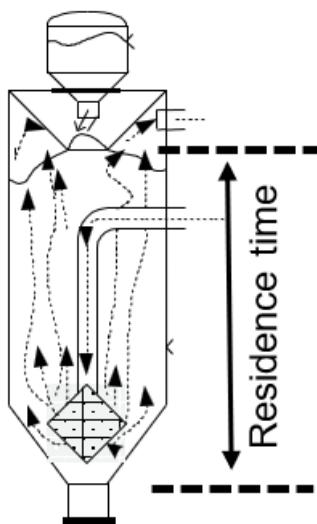
- the drying time in hours
- the bulk density of the plastic resin in lb/ft³ or kg/m³

The amount of material consumed will be determined by the process. For example, a molder that uses a 14-second cycle to injects parts and runners with a total weight of 53 grams will have a consumption of:

$$\text{Consumption} = 53\text{g} / 14\text{s} = 3.79 \text{ g/s} = \mathbf{13.6 \text{ kg/hr}} = \mathbf{30 \text{ lb/hr}}$$

Drying time

The drying time is dictated by the material. For example, a polycarbonate where the manufacturer specifies 4 hours of drying means that the material must reside more than four hours inside the drying hopper before being molded. In other words, the material that enters from the top of the hopper must stay inside at least 4 hours before being injected. Drying time considers the time the material resides inside the drying hopper.



V-4. Volume/residence time

Bulk density

Bulk density is a value provided by the resin supplier and represents the space that the resin occupies in its granular state. It should not be confused with the density of the material after molding; remember that bulk density considers the empty spaces between resin pellets (granules). The most common bulk density units are kg/m^3 or lb/ft^3 . For example, using a polycarbonate with a density of 52 lb/ft^3 (833 kg/m^3), a filled

container that holds one cubic foot would weigh 52 lb, and a filled container that holds one cubic meter would weigh 833 kg.

Example:

A molding machine consumes 30 lb/hr (13.5 kg/hr) of polycarbonate, which has a bulk density of 52 lb/ft³ (833 kg/m³) and requires drying for 4 hours.

Using the equation and substituting:

$$\text{Volume} = \text{consumption} \times \frac{\text{drying time}}{\text{bulk density}}$$

$$V = 30(\text{lb/hr})4(\text{hr}) / 52(\text{lb/ft}^3) = 2.3 \text{ ft}^3$$

We find that this process requires a drying hopper equal or slightly larger than 2.3 ft³ (65 liters).

If the process uses reprocessed material or regrind, we must take into consideration that the regrind has another bulk density, since it is not in the same granular form as the virgin material. In addition to the bulk density of the regrind, we would need to know the proportion of virgin material to regrind.

Consider the example above, with 20% regrind:

- consumption = 30 lb/hr, 80% virgin and 20% regrind
- virgin material bulk density = 52 lb/ft³
- regrind bulk density = 36 lb/ft³
- drying time = 4 hours

Using the following equation:

$$\text{Volume} = T * C * \left(\frac{\%V}{D_{\text{virgin}}} + \frac{\%R}{D_{\text{regrind}}} \right)$$

Where:

T = drying time (hours)

C = resin consumption (lb/hr or kg/hr)

D_{virgin} = virgin material density (lb/ft³ or kg/m³)

$D_{regrind}$ = regrind density (lb/ft³ or kg/m³)

%V = % of virgin

%R = % of regrind

Replacing the values:

$$\begin{aligned}\text{Volume} &= 4 \text{ hours} \times 30 \text{ lb/hr} [0.8 \times 52 \text{ lb/ft}^3 + 0.2 \div 36 \text{ lb/ft}^3] \\ &= 2.51 \text{ ft}^3\end{aligned}$$

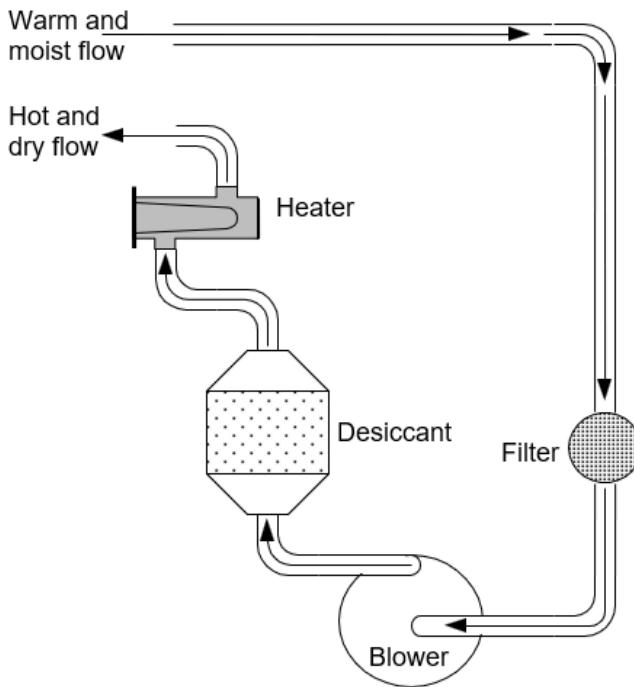
We would need a hopper equal or slightly larger than 2.51 ft³. This volume is larger than the example with 100% virgin resin (2.3 ft³). This is to be expected as regrind takes up more space than virgin material. Remember that the shape of the regrind is a mixture of flakes, shavings, and dust, and that the virgin material consists of compact granules.

Resin loader

The purpose of a resin loader is to keep the drying hopper full of material. The drying hopper must always be full of resin. If the hopper is accidentally emptied halfway, for example, and then filled it is likely that the material that was filled will not be completely dry at the time of molding, since the material did not reside in the dryer the required amount of time.

Dryer

The dryer consists of a filter, a blower, a desiccant bed, and a heater in the drying circuit.



V-5. Drying circuit

The dryer must guarantee a hot and dry air flow. Humid air returning from the drying hopper is dried and heated before being returned to the hopper.

Filter

The humid air returning from the drying hopper passes through a filter, preventing plastic dust from contaminating the desiccant. It is extremely important that the filter element is regularly examined, kept clean and replaced according to the manufacturer's specifications. A stopped-up filter obstructs air flow and consequently will reduce the drying capacity of the dryer. In addition, a perforated filter will allow the passage of plastic particulates to the desiccant material and will damage it.

Blower

Humid, warm, filtered air returns to the blower. The blower is responsible for maintaining a constant flow of air throughout the system. Remember

that each process requires a specific air flow that is a function of the type of material and material consumption.

The maximum flow factor is:

$$\begin{aligned}1 \text{ cfm} &\square 1 \text{ lb/hr of material} \\63 \text{ liters/min} &\square 1 \text{ kg/hr of material}\end{aligned}$$

This is the maximum flow factor; consult your resin supplier to obtain the actual value for your material.

For example:

Brand X polycarbonate is used in a process that consumes 50 lb/hr.

$$\text{Flow} = \text{material flow factor} \times \text{consumption}$$

If we do not have the manufacturer's recommended flow factor, we assume a factor of 1 cfm/lb/hr and calculate:

$$\text{Flow} = (1 \text{ cfm/lb/hr}) \times (50 \text{ lb/hr}) = \mathbf{50 \text{ cfm}}$$

Now, if we know the flow factor, let's assume it is 0.95 cfm/lb/hr, we get:

$$\text{Flow} = (0.95 \text{ cfm/lb/hr}) \times (50 \text{ lb/hr}) = \mathbf{47.5 \text{ cfm}}$$

The maximum flow factor is safer, but it is not real. If you are confirming an existing process and want a quick verification value, use the maximum factor. If, during a quick verification, the calculated flow is less than the flow that the existing dryer is capable of, then the dryer is adequate for your process. Now, if you are sizing a dryer for a process, make your calculations with the actual flow factor.

Desiccant bed

The desiccant bed contains a drying material that removes moisture from the air. These beds are pre-installed in a suitably sized dryer. The equipment arrives with the proper desiccant system for that equipment.

Now, it is important to verify that the humid air entering the desiccant bed has a temperature less than 65°C (150°F), because if the humid air returning to the dryer is above that, the desiccant **will not** work. If humid

air returns to the dryer above 65°C (150°F), an aftercooler must be added to the system to reduce the return air temperature.

When selecting a dryer, check whether you need an aftercooler in the dryer's humid air return line. It was mentioned above that it is assumed that about 56°C (100°F) temperature is lost during drying. Based on this information select the material with the highest drying temperature that you consider using in that dryer and subtract 56°C (100°F). This way you will know if the return of moist air is above 65°C (150°F).

For example: with Brand XY PET, the manufacturer specifies that it should be dried to 160°C (320°F). Is a heat exchanger required? If we subtract 56°C from the drying temperature, we get:

$$\text{Return temperature} = 160^\circ\text{C} - 56^\circ\text{C} = \mathbf{104^\circ\text{C}}$$

Since 104°C is greater than 65°C, this system will require a heat exchanger.

If you have noticed that 100°F is not equal 56°C, it is not a mistake; we are talking about Delta temperature change. When a Delta change is converted from Fahrenheit to Celsius, the change in Fahrenheit is multiplied by 5/9. If the temperature change is from Celsius to Fahrenheit, the change in Celsius is multiplied by 9/5. This conversion is only applied when it has to do with a Delta change in temperature.

Heater

Finally, the dry air coming out of the desiccant bed is heated to the temperature required by the material and is sent to the drying hopper. The heater will turn on automatically, maintaining the temperature set by the equipment technician. No calculations need to be done to determine the heater's size; fortunately, the equipment arrives with the right heater for its dryer.

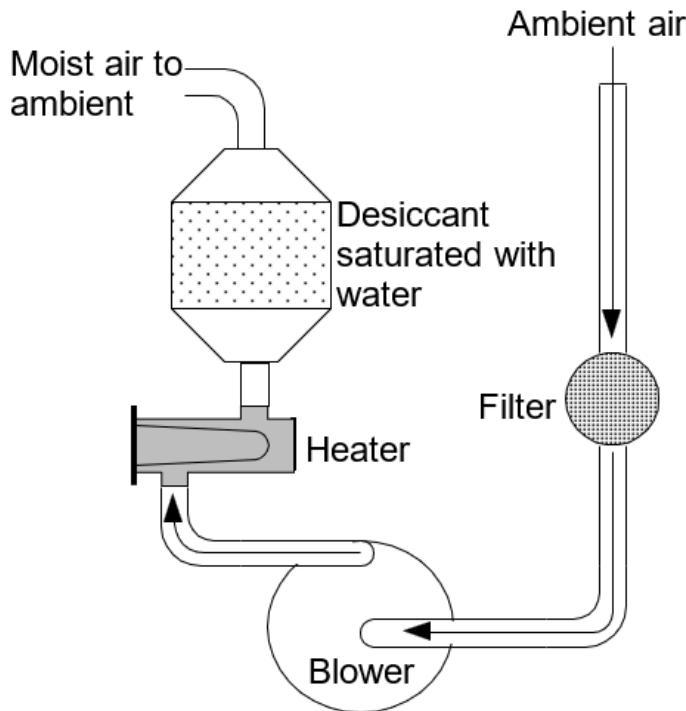
Once the desiccant is saturated with water, it goes through a regeneration process.

Regeneration

The desiccant cannot remove humidity forever; eventually it will become saturated with water. In the regeneration stage, the saturated bed is

automatically changed for a dry one and then goes through a regeneration stage.

The components of the regeneration circuit are the ambient air inlet, the particulate filter, the blower, the regeneration heater, the saturated desiccant bed, and the discharge to the atmosphere.



V-6. Regeneration stage

In this stage of regeneration, air that is heated over 200°C (400°F) is forced through the desiccant material, removing moisture and discharging it to the environment. Because of this, the filters should always be kept clean and without perforations, in order to prevent plastic dust from reaching the desiccant. The regeneration air is so hot that it can melt most plastics, and that melt could cover the desiccant and damage it. Once dried, the desiccant bed will enter a cooling stage, waiting to take the place of the other bed once that one becomes saturated.

Even though the equipment performs this operation automatically, the method depends on the equipment manufacturer. Some come with a fixed time in the system's control. Other controls do this by measuring the ambient discharge temperature; when the discharge temperature rises to a certain level it indicates that the hot air flow has stopped removing moisture.

The equipment manufacturer will recommend how often the desiccant material should be replaced. Many recommend that it be changed annually, others recommend replacing only after the dewpoint goes above -30°C (-20°F).

Although an excellent tool, dewpoint meters are optional in some equipment. If your dryer is not equipped with a dewpoint meter, consider purchasing a portable unit.

The switching of desiccant beds in order to regenerate varies with the type of dryer. It can be done by means of valves that change the direction of flow, by means of desiccant beds that rotate in a carousel, by means of a solid desiccant wheel that rotates continuously, etc.

A unit operating with a dewpoint of -40°C (-40°F) means that the air flowing through the dryer is super dry; however, it is no guarantee that the material is ready to be molded.

There exists equipment that measures the water content in the resin. This basically operates by comparing the weight of the resin before and after evaporating the water content with heat.

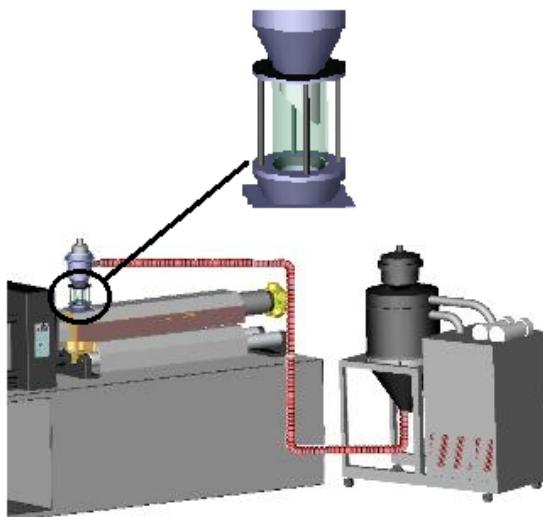
In addition, there are dryers that do not use desiccant material and only operate with heat; their use is limited to non-hygrosopic materials. Others operate by combining hot air and negative pressure. Let me explain, decreasing the pressure in a sealed vessel reduces the evaporation temperature of water and improves the extraction of moisture from the material. Verify with your resin supplier what the best dryer is for your application.

Setting up drying systems

There are different drying system configurations: portable unit, hopper above an extruder, integrated unit, and central drying system.

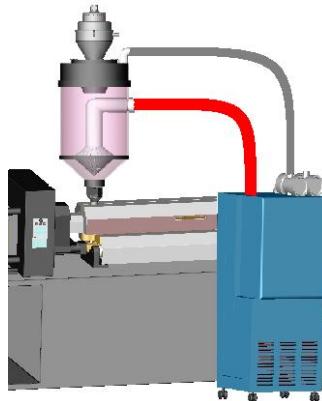
Portable unit

The portable unit offers the advantage that its use does not necessarily have to be dedicated to a single extruder. The loader that transports the material to the injection unit will suction a restricted amount of dried material. Remember that once the material is removed from the dryer, the material is ready to absorb moisture.



V-7. Portable unit

Hopper above the extruder

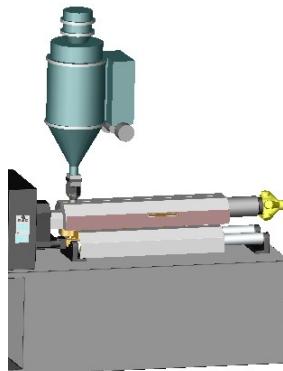


V-8. Hopper above the extruder

Installing the hopper above the extruder reduces the risk of the material absorbing moisture from the atmosphere, since the dried material goes directly from the drying hopper to the extruder.

Integrated unit

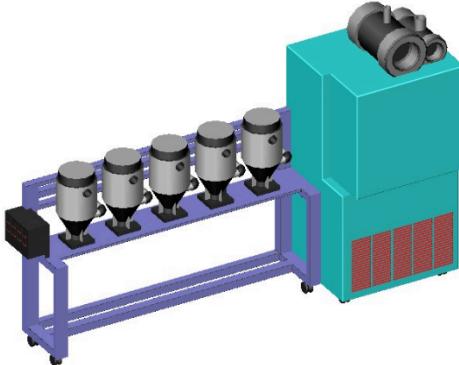
The integrated unit has the dryer and drying hopper assembled together. This system is mounted on the injection unit and, like the hopper on the extruder, reduces the risk of the material absorbing moisture from the atmosphere. Its major advantage is that it does not take up space on the factory floor.



V-9. Integrated unit

Central drying system

The central drying system connects multiple drying hoppers to a common dryer.



V-10. *Central*
The biggest
it can be in a
area for multiple
system facilitates
changing, takes
and confines the
specific place.

drying system
advantage is that
centrally located
machines. This
material
up little space,
drying area to a

For each mold using hygroscopic thermoplastic, the corresponding dryer equipment must be determined. Look at the exercise below.

Example:

material \square Polycarbonate

consumption \square 50 lb/hr, 80% virgin and 20% regrind

Use the tables below:

- *Materials and their drying parameters*
- *Materials and their bulk densities*
- *Dry air flow required*

density ρ of virgin material = 40 lb/ft³

density ρ of regrind material = 36 lb/ft³

$T = 250^{\circ}\text{F}$ to 270°F

$t_s = 4 \text{ hr}$

flow rate = $0.95 \text{ (ft}^3/\text{min}) /(\text{lb}/\text{hr})$

(Always consult the material manufacturer)

Determining the dryer hopper:

$$\text{Volume} = t_s \times \text{consumption} (\% \text{virgin} \div \rho_{\text{virgin}} + \% \text{regrind} \div \rho_{\text{regrind}})$$

$$= 4 \times 50(0.8 \div 40 + 0.2 \div 36) = \mathbf{5.1 \text{ ft}^3}$$

Dryer Size:

Flow = rate x consumption = $0.95 \times 50 = \mathbf{47.5 \text{ ft}^3/\text{min}}$

Recommended system:

- dryer with a minimum flow of $47.5 \text{ ft}^3/\text{min}$
- hopper with a minimum size of 5.1 ft^3
- drying temperature of 250°F
- aftercooler to reduce return temperature

Common Name	Description	Hygroscopic?	Drying Hours	Drying Temp °F
ABS/PVC	ABS/PVC Alloy	Yes	2-3	160 - 170
ABS (Molding Grade)	Acrylonitrile-Butadiene-Styrene Thermopolymer	Yes	2-4	190 - 220
ABS/PC	ABS/Polycarbonate Alloy	Yes	4-5	220 - 230
Acetal (copolymer)	Acetal Resin	Yes	2-3	200 - 220
Acetal (homopolymer)	Acetal Resin	No	1-2	200 - 220
Acrylic	Methyl Methacrylate	Yes	2-3	170 - 190
CA (Acetate)	Cellulose Acetate	Yes	2-3	160 - 180
CAB (Butyrate)	Cellulose Acetate/Butyrate	Yes	2-3	160 - 180
CAP (Propionate)	Cellulose Acetate/Propionate	Yes	2-3	160 - 180
EVOH	Ethylene-Vinyl Alcohol Copolymer	Yes	2-3	195 - 225
HDPE	High Density (Linear) Polyethylene	No	1-2	160 - 180
HDPE w/max 3% black	High Density (Linear) Polyethylene	Yes	3-4	160 - 180
HDPE w/max 4% black	High Density (Linear) Polyethylene	Yes	4-5	160 - 180
Ionomer	Ionomer Resin	Yes	7-8	150 - 160
LCP	Liquid Crystal Polymer (Aromatic Polyester)	Yes	3-4	300 - 310
LDPE	Low Density (Conventional Polyethylene)	No	1-2	160 - 180
LDPE w/max 3% black	Low Density (Conventional Polyethylene)	Yes	3-4	160 - 180
LDPE w/max 40% black	Low Density (Conventional Polyethylene)	Yes	4-5	160 - 180
Nitrile	Acrylonitrile Terpolymer	Yes	5-6	160 - 180
Nylon 6, 6/6, 6/12	Crystalline Nylon (Caprolactan)	Yes	5-6	160 - 180
Nylon (Amorphous)	Super Tough Nylon	Yes	4-5	180 - 190
Nylon (Transparent)	Transparent Nylon	Yes	4-5	180 - 190
OSA	Olefin-Modified Styrene-Acrylonitrile Copolymer	Yes	2-3	180 - 190
PBT	Polybutylene-Terephthalate Copolymer	Yes	2-3	250 - 270
PBT/PET	PBT/PET Alloy	Yes	4-5	350 - 370
PC	Polycarbonate	Yes	3-4	250 - 270

V-11. Table of materials and their drying parameters

Common Name	Description	Hygroscopic?	Drying Hours	Drying Temp °F
PC/PBT/E	Polycarbonate/PBT/ Elastomer Alloy	Yes	3-4	220-230
PCS	Polycarbonate-Styrene Copolymer	Yes	2-3	220-230
PCTA	Cyclohexane-Terephthalate Copolymer	Yes	3-4	160-180
PEEK	Polyetheretherketone	Yes	3-4	300-320
PEM	Polyetherimide	Yes	6-7	300-310
PES	Polyethersulfone	Yes	3-4	300-320
PET (Molding Grade)	Polyethylene- Terephthalate (Polyester)	Yes	2-4	250-270
PETG	Amorphous PET Copolymer	Yes	3-4	140-150
Polyarylate	Amorphous Aromatic Polyester	Yes	5-6	250-260
Polysulfone	Polyether, Polyarylsulfone	Yes	4-5	250-260
Polyurethane	Polyurethane Elastomer	Yes	2-3	180-200
PP	Polypropylene	No	1-2	170-190
PPA	Polyphthalamide	Yes	6-7	175-180
PPC	Polyphthalate Carbonate	Yes	3-4	260-270
PPO	Polyphenylene	Yes	2-4	200-250
PPS	Polyphenylene Sulfide	Yes	3-4	280-290
PPS (40% Glass)	Polyphenylene Sulfide	Yes	3-4	300-320
PS (Styrene)	Polystyrene	No	1-2	180-190
PTMT	Polytetramethylene- Terephthalate	Yes	2-3	210-220
PVC (Flexible)	Polyvinyl-Chloride	No	1-2	160-180
PVC (Rigid)	Polyvinyl-Chloride	No	1-2	160-180
SAN	Styrene-Acrylonitrile	Yes	3-4	180-190
SAN (Modified)	Styrene-Acrylonitrile (with Olefin Elastomer)	Yes	3-6	160-180
SMA	Styrene-Maleic Anhydride	Yes	2-3	200-210
TPE	Thermoplastics Polyester	Yes	2-3	210-220
TPR	Thermoplastic Rubber	Yes	2-3	150-170
XT	Impact-Modified Acrylic Resin	Yes	3-4	170-190

V-IIa. Table of materials and their drying parameters (cont.)

Note: These values are a guide, always consult with the material manufacturer.

Common Name	Description	Density lb/ft³ Virgin	Density lb/ft³ Grind
PC/PBT/E	Polycarbonate/PBT/Elastomer Alloy	42	38
PCS	Polycarbonate-Styrene Copolymer	38	34
PCTA	Cyclohexane-Terephthalate Copolymer	52	44
PEEK	Polyetheretherketone	52	44
PEM	Polyetherimide	52	46
PES	Polyethersulfone	52	46
PET (molding grade)	Polyethylene Terephthalate (Polyester)	54	46
PETG	Amorphous PET Copolymer	50	40
Polyarylate	Amorphous Aromatic Polyester	50	44
Polysulfone	Polyether, Polyarylsulfone	50	44
Polyurethane	Polyurethane Elastomer	48	42
PP	Polypropylene	33	27
PPA	Polyphthalamide	48	40
PPC	Polyphthalate Carbonate	50	44
PPO	Polyphenylene	50	44
PPS	Polyphenylene Sulfide	50	44
PPS (40% glass)	Polyphenylene Sulfide	50	44
PS (Styrene)	Polystyrene	35	27
PTMT	Polytetramethylene Terephthalate	50	44
PVC (Flexible)	Polyvinyl Chloride	48	34
PVC (Rigid)	Polyvinyl Chloride	50	32
SAN	Sytrene Acrylonitrile	40	34
SAN (Modified)	Styrene Acrylonitrile (with Olefin Elastomer)	42	36
SMA	Styrene-Maleic Anhydride	38	32
TPE	Thermoplastic Polymer	48	42
TPR	Thermoplastic Rubber	48	42
XT	Impact-modified Acrylic Resin	40	36

V-12. Table of materials and their bulk densities

V-12a. Material table and their bulk densities (cont.)

Common Name	Description	Density lb/ft³ Virgin	Density lb/ft³ Regrind
ABS/PVC	ABS/PVC Alloy	40	32
Acetal (copolymer)	Acetal Resin	40	35
Acetal (homopolymer)	Acetal Resin	40	32
Acrylic	Methyl Methacrylate	42	36
CA (Acetate)	Cellulose Acetate	38	32
CAB (Butyrate)	Cellulose Acetate/Butyrate	39	33
CAP (Propionate)	Cellulose Acetate/Propionate	40	34
EVOH	Ethylene-Vinyl Alcohol Copolymer	36	32
HDPE	High Density (Linear) Polyethylene	35	28
HDPE w/max 3% black	High Density (Linear) Polyethylene	34	26
HDPE w/max 4% black	High Density (Linear) Polyethylene	34	26
Ionomer	Ionomer Resin	44	36
LCP	Liquid Crystal Polymer (Aromatic Polyester)	50	46
LDPE	Low Density (Conventional Polyethylene)	32	24
LDPE w/max 3% black	Low Density (Conventional Polyethylene)	32	24
LDPE w/max 40% black	Low Density (Conventional Polyethylene)	32	24
Nitrile	Acrylonitrile Terpolymer	40	32
Nylon 6, 6/6, 6/12	Crystalline Nylon (Caprolactan)	41	35
Nylon (Amorphous)	Super Tough Nylon	42	36
Nylon (Transparent)	Transparent Nylon	41	35
OSA	Olefin-Modified Styrene-Acrylonitrile Copolymer	42	36
PBT	Polybutylene-Terephthalate Copolymer	52	44
PBT/PET	PBT/PET Alloy	50	44
PC	Polycarbonate	40	36

Note: These values are a guide, always consult with the material manufacturer.

Material	ft³/min per lb/hr
ABS	0.75
Acetal	0.80
Acrylic	0.95
PA (Nylon)	0.90
PBT	0.80
PC	0.95
PET	1.00
PPO	0.80
SAN	0.80
TPE	0.80
Acetal (homopolymer)	0.75
HDPE	0.75
LDPE	0.75
Polypropylene	0.75
Polystyrene	0.75
PVC	0.75

V-13. Dry air flow required for each lb/hr of material consumption

Note: These values are a guide, always consult with the material manufacturer.

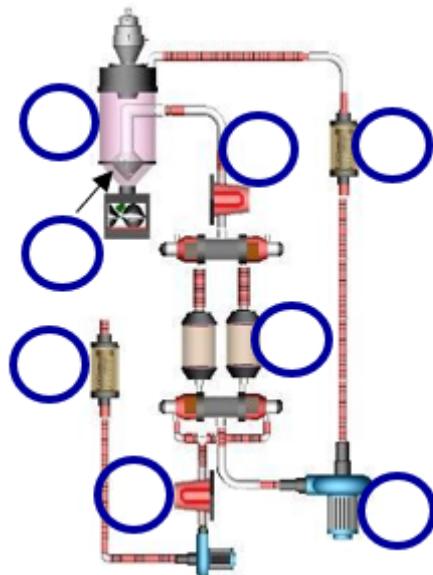
Questions

- 1) What are the four factors that govern drying?
 - a. drying time, oil pressure, air flow, and drying temperature
 - b. drying time, super-dry air, air flow, and drying temperature
 - c. drying time, humid air, air flow, and drying temperature
- 2) In the drying of hygroscopic material, the dewpoint is
 - a. the temperature at which the material is dried
 - b. ambient temperature
 - c. the temperature at which water, which exists in the drying flow, condenses
- 3) Select all the correct sentences:
 - a. Drying temperature is specific for each material.
 - b. Drying temperature is the temperature at which the water vapor condenses.
 - c. The temperature at which humidity condenses is also known as dewpoint.
 - d. Drying flow is set to a standard 15 cfm for all types of commercial dryers.
 - e. Each material manufacturer specifies variables such as drying temperature, time, and flow.
- 4) Only hygroscopic materials are dried.
 - a. True, only hygroscopic materials that require high clarity are dried.
 - b. False, some materials that require high clarity are dried in order to maintain their translucence.
- 5) Bulk density considers gaps or spaces between granules or pellets.
 - a. True.
 - b. False, the space is negligible.
- 6) A material being used at the rate of 50 lb/hr requires drying for 4 hours at a temperature of 265°F. The bulk density of the material is 40lb/ft³. The dryer must have a volume of at least:
 - a. $40 \text{ lb/hr} \times 4 = 160 \text{ ft}^3$
 - b. $50 \text{ lb/hr} \times 4 \text{ hours} / 40 \text{ lb/ft}^3 = 5 \text{ ft}^3$

c. 265 ft³

- 7) A material requires drying for 4 hours in a 30 lb/hr process. Considering that 80% is virgin material with a bulk density of 40 lb/ft³ and 20% is regrind with a bulk density of 36 lb/ft³, determine the minimum volume of the drying hopper.
- volume = 4 hours x 30 lb/hr x 36 lb/ft³
 - volume = 4 hours x 30 lb/hr x 40 lb/ft³
 - volume = 4 hours x 30 lb/hr [0.8 x 40 lb/ft³ + 0.2 x 36 lb/ft³]
- 8) A material requiring a drying temperature of 260°F does not require the use of an aftercooler on the return line.
- True, if we subtract the loss of 100°F during drying, we will have a return of 160°F and will not require an aftercooler.
 - False, if we subtract the loss of 100°F during drying, we will have a return to 160°F (>150°F) and will require an aftercooler.
- 9) If the dryer hopper is too small, it is recommended to increase the actual drying temperature until it is higher than the recommended drying temperature.
- True, this is a good practice.
 - False, this is not a recommended practice.
- 10) Under normal conditions, the dryer hopper must be kept full of material.
- True.
 - False.
- 11) To determine whether you need to purchase an aftercooler for the return of your dryer, how much do you subtract from the drying temperature?
- 100°F or 56°C.
 - 38°F.
 - 56°F or 100°C.
- 12) To determine the size of the dryer it is necessary to know
- the dry air flow, temperature of the dryer, and bulk density of the plastic resin.
 - the material consumption, drying time, and bulk density of the plastic resin.
 - the dry air flow, return temperature, and the size of the heater.

- 13) It is extremely important that the filter elements are examined and cleaned
- every two months.
 - according to the maintenance specifications indicated by the equipment manufacturer.
- 14) A hygroscopic thermoplastic with a flow rate of 0.8 cfm/lb/hr needs to be dried for a process of 50 lb/hr. Determine the flow of the dryer.
- minimum flow = $50 \text{ lb/hr} \times 0.8 \text{ cfm/lb/hr}$
 - minimum flow = 50 cfm
 - minimum flow = 50 liters per minute
- 15) Write the number of each description to its corresponding place in the drawing.
1. drying hopper
 2. diffuser
 3. drying circuit blower
 4. drying circuit filter
 5. desiccant beds
 6. regeneration heater



Blending and Material Handling

Automatic dosing and mixing of plastic resin is a must. The high cost of resins, additives and labor makes automation an economically viable option. If you add to this the losses as a result of manual handling, automation could have a return on investment of less than one year.

Some of these deficiencies are:

- Manual blending of virgin resin and pigment could result in an excess of 0.5%. This waste can exceed the investment in an automatic blender in only one month.
- Handling premixed material increases storage costs.
- Losses can occur as a result of resin contamination during manual mixing.
- Manual handling takes up more space on the production floor.

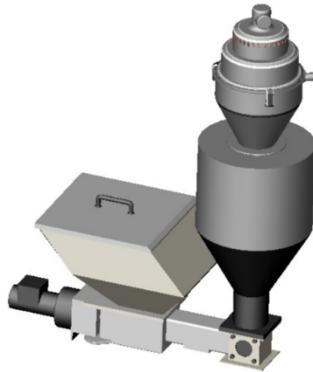
Types of additive feeders:

- direct and volumetric feeder
- gravimetric feeder
- pneumatic proportional valve

Direct and volumetric feeder

This type of feeding is done in the throat of the extruder. The components of this type of feeder are the additive hopper, motor, and auger. The auger rotates, feeding the dose of additive directly into the flow of resin. This is also known as volumetric feeding.

The feeder is installed between the hopper loader and the extruder's throat. The unpigmented resin flows by means of gravity. Feeding is a function of the auger's revolutions per minute (rpm); its ratio depends on the type of ingredient being added. For example, granular pigment dosing amounts fluctuate from 0.25% to 6%.



V-14. Additive feeder

It is important to properly size your equipment; you will need to take into consideration:

- process' total consumption, in kg/hr or in lb/hr
- type of materials
- proportion (dose) of the additive
- recovery time
- whether material flows easily or clumps

Continuous feeding is normally for extrusion processes and periodic feeding is for injection. Periodic feeds only happen during recovery. Remember that the injection unit uses material only during the recovery stage; the rest of the time the unit is not consuming material.

Calculating periodic feeding consumption:

Think of an application where you want to add colorant at a ratio of 2.5% in the injection process. The total injection weight (parts with runner) is 0.12 kg, and recovery lasts 3 seconds. Remember that dosing happens during the recovery period.

$$\begin{aligned} \text{Recovery consumption} &= \\ (\text{injection shot weight}) / (\text{recovery time}) & \\ = (0.12 \text{ kg}) / (3 \text{ s}) &= 0.04 \text{ kg/s} = \mathbf{144 \text{ kg/hr (317 lb/hr)}} \end{aligned}$$

So, the feeding consumption would be:

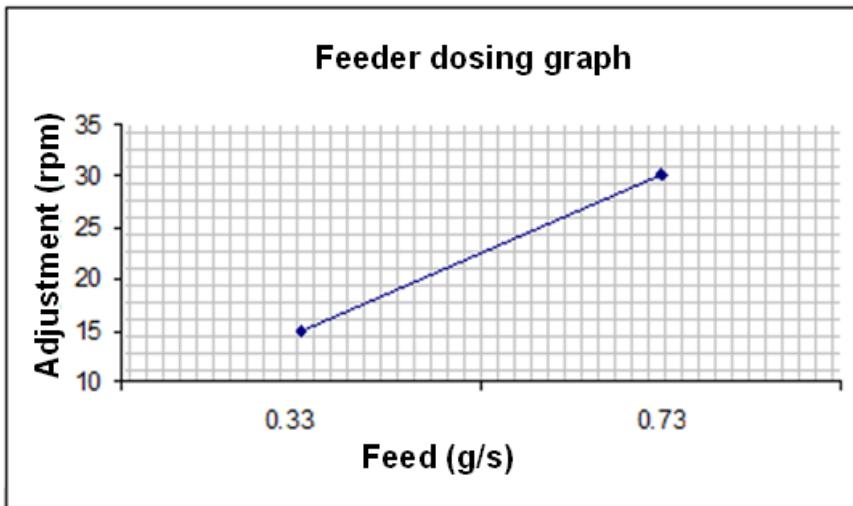
$$\begin{aligned} &= (\text{total consumption}) \times (\text{feeding ratio}) \\ &= 144 \text{ kg/hr} \times 0.025 \\ &= \mathbf{3.6 \text{ kg/hr (7.9 lb/hr)}} \end{aligned}$$

Although it may seem simple, it is common to see a feeder being sized for the total cycle time, when it should be done only for the recovery time.

This equipment provides an access from which sampling is done at distinct feeding speeds at a defined time, then is weighed to calculate:

$$\text{Feeding} = (\text{weight of sample}) / (\text{feeding time})$$

This is used to create a graph of speed adjustment versus feeding rate.

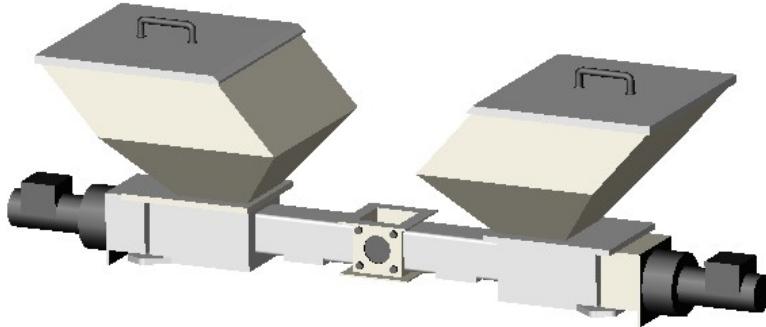


V-15. Graph of speed adjustment vs. feeding rate

This graph could change with a change in pellet size and geometry. You should also create a calibration table for each colorant.

These systems could have a deviation of $\pm 0.2\%$. Now if you want better precision, consider a gravimetric blending system.

If you want to feed more than one ingredient you can attach more than one additive feeder to a single system; for example, one for dosing pigment and another for dosing regrind.



V-16. Two feeders mounted in one system

Now, feeding three or more additives is usually accomplished with gravimetric feeding systems.

There are materials that do not flow easily and need to be handled by special feeding equipment. For example, a very lightweight regrind might require an agitator to help it flow. Consult your equipment supplier or send a sample of your material if you understand that it does not flow easily.

Gravimetric feeder

These systems are distinguished by their ability to weigh the ingredients. This makes them more precise and can include multiple functions, such as: weighing ingredients, mixing, programmable recipes, inventory control, and dosing with multiple ingredients.

There are products that require multiple additives; these can be regrind, recycled, colorant, softeners, clarifiers, lubricants, fibers, etc.

A gravimetric system is typically composed of material hoppers, feeder systems, weighing hopper, mixing chamber, and discharge outlet.



V-17. Gravimetric system

Although their operations vary according to the manufacturer, they conceptually function in a similar way. Each hopper is filled with an ingredient, such as virgin, regrind, or granulated colorant. By means of a slide gate or an auger, the ingredients are alternately fed to a weighing hopper. Once the proper proportion is weighed, it is transferred to a mixing chamber. After being mixed, it is passed to a discharge hopper for later processing.

To determine the appropriate equipment, you must know the total material consumption in kg/hr or lb/hr, the type of materials, the proportion of each additive, and whether the materials flow easily or if they clump.

For example, there is an application where you want to mix 3 ingredients, virgin, regrind and a colorant at a ratio of 77.5%, 20% and 2.5%, respectively. This process consumes 1 kg per minute. The total consumption is $1 \text{ kg/min} = 60 \text{ kg/hr (132 lb/hr)}$. Therefore, the additive consumption of each ingredient would be:

$$\text{Virgin} = 60 \text{ kg/hr} \times 0.775 = \mathbf{46.5 \text{ kg/hr (102.51 lb/hr)}}$$

$$\text{Regrind} = 60 \text{ kg/hr} \times 0.2 = \mathbf{12 \text{ kg/hr (26.46 lb/hr)}}$$

$$\text{Colorant} = 60 \text{ kg/hr} \times 0.025 = \mathbf{1.5 \text{ kg/hr (3.3 lb/hr)}}$$

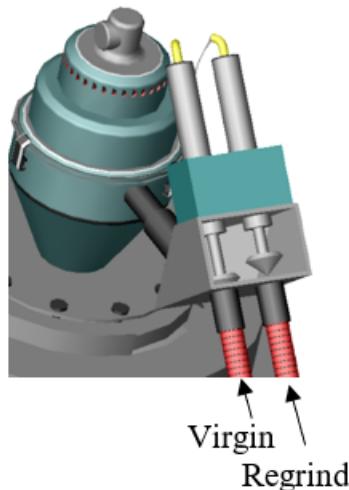
These calculations are not necessarily required on modern equipment, just providing the percentages will be enough. Now if you are purchasing new equipment, those calculations are required to configure it. Talk to the equipment supplier; they can do the calculations for you.

Remember, it is important to indicate if the material or materials do not flow easily. Consult your equipment supplier or send samples of your material if you understand that your material does not flow well.

Pneumatic proportional valve

The simplest of mixers is the pneumatic proportional valve. This is mostly used in the mixing of virgin with regrind, where a variation in proportion of up to 5% is acceptable.

It works by means of valves with rubber plungers that control the suction of one material at a time.



V-18. Pneumatic proportional valve

The proportional valve is mounted on the hopper loader; while the hopper suctions the material, the valve proportionally controls the amount of

material that enters the hopper loader. This is done by means of pneumatic cylinders that control the opening of the plungers.

For example, materials A and B must be proportionally mixed at a ratio of 30% and 70%, respectively, in a vacuum adjusted to suction for 40 seconds.

The set times could be:

Time for material A = **3 seconds**

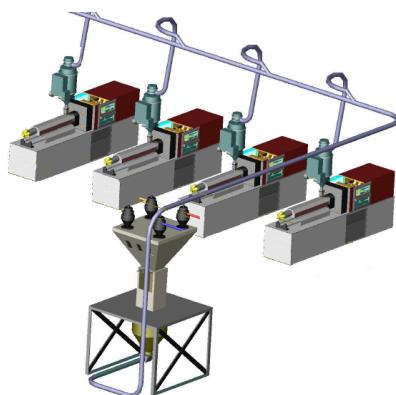
Time for material B = **7 seconds**

Properly adjusted, this would create 8 layers alternating between materials A and B.

The advantages of proportional valves are that they are economical and easy to install. Their biggest disadvantage is that they are not proportionally accurate. Because of this, it is not recommended for mixtures where the variation of the required dosage is less than 5%, as would be the case of colorant.

Central blending system

This type of dosing and mixing is used when the equipment is shared by multiple processes. Consult an automatic blending and mixing design specialist before purchasing an automatic mixing system.



V-19. Central blending system

Questions

- 1) What information should be known before purchasing equipment to blend colorant?
 - a. Pigment color, colorant manufacturer's email, and material consumption in kg/hr.
 - b. Total process consumption (in kg/hr or lb/hr), type of material, if it flows easily or clumps, proportion of additive, and type of dosing (continuous or periodic).
 - c. The cost of the additive feeder.
- 2) An injection process requires 2% colorant. The total injection weight (parts with runner) is 0.12 kg, recovery lasts 3 seconds, and the process produces parts every 18 seconds. Determine the feeding consumption.
 - a. Consumption = $(0.12 \text{ kg}/3\text{s}) \times 0.02$
 - b. Consumption = $(0.12 \text{ kg}/18\text{s}) \times 0.02$
 - c. Let the colorant supplier calculate it for you.

Controlling the Water Temperature to the Mold

It is important to identify the needs of your process and meet those needs with well-thought-out solutions, so:

- Before molding, or making any adjustments to the injection machine, some initial calculations should be made by what is known as “molding from the desk”.
- Remember that you are working with expensive equipment; do not rush the job.

In this part we will talk about material consumption, heat removed, water flow, cooling time, and temperature control.

Material Consumption

To determine the material consumption of a particular mold you should know the approximate cycle time of the molding process in seconds and the amount of material required by the mold, in grams or in the unit of your choice.

The molding cycle time and the amount of material can be provided by the product designer or mold manufacturer. Material consumption is obtained with the following equation:

$$\text{Material consumption} = \frac{\text{total amount of material}}{\text{process cycle time}}$$

For example:

With a mold that molds polycarbonate components, the total weight required to fill the mold is 275 grams and the expected mold cycle is 9 seconds.

$$\text{Material consumption} = 275 \text{ grams} / 9 \text{ seconds} = \mathbf{30.6 \text{ gr/s}}$$

This means that this process will consume about 31 grams/second (246 lb/hr or 112 kg/hr).

Heat removed

Heat removed is the amount of heat removed per mass unit in a specific thermoplastic. This is normally measured in units of BTU/hr, kW, and cooling tons, where:

- kW = 3415.18 BTU/hr
- cooling ton for a chiller = 12000 BTU/hr
- cooling ton for a tower = 15000 BTU/hr

$$\text{Heat removed} = \frac{(\text{total quantity of material}) * (\text{energy required for the material})}{(\text{process cycle time})}$$

where the amount of material it takes to fill the mold is in grams, the energy required for the material is in Joules/gram, and the process cycle time is in seconds. The table below shows the energy required for certain materials.

Energy required for each material		
Material	Energy required	
	Joules/gram	BTU/lb
PC	368	158
ABS	369	159
PS	394	169
LDPE	572	246
HDPE	801	344
PVC	434	187
PA66	615	264
PP	670	288
PET	283	122

V-20. Table of energy required for some materials

The values provided in this table are a reference; get the actual values from your resin supplier.

Example:

A mold is used to make components in polycarbonate, with a total weight of 275 grams required to fill a mold with an expected cycle of 9 seconds. How much heat would be consumed?

The previous table lists polycarbonate as requiring 368 Joules/gr of energy.

$$\text{Heat removed} = \frac{(\text{total quantity of material}) * (\text{energy required for the material})}{(\text{process cycle time})}$$

$$= 11244 \text{ watts} = 11.24 \text{ kW} = \mathbf{38400 \text{ BTU/hr}}$$

$$= \frac{(275 \text{ g}) \times (368 \text{ Joules/g})}{9 \text{ s}} = \mathbf{11244 \text{ J/s}}$$

This means that to cool this mold, it would consume 11.24 kW or 38400 BTU/hr from your cooling system.

Another method utilized is the use of tables provided by the cooling equipment manufacturers.

Material	kg/hr/ton	lb/hr/ton
HDPE	14	30
LDPE	16	35
PP	16	35
NYLON/PA	18	40
PET	18	40
PS	23	50
ABS	23	50
PVC	30	65
PC	30	65
PETG	11.5	25

V-21. Table of thermal load of some materials

This method is the most used, as it is simpler, and it works.

Let's again use the example above, where a mold for polycarbonate components has a total weight of 275 grams and an expected cycle of 9 seconds.

According to the table, the energy factor for PC is 30 kg/hr/ton or 65 lb/hr/ton.

$$\text{Material consumption} = \frac{\text{total amount of material}}{\text{process cycle time}}$$

$$= 275 \text{ g} / 9 \text{ s} = \mathbf{30.6 \text{ g/s}}$$
$$110 \text{ kg/hr} = \mathbf{242 \text{ lb/hr}}$$

$$\text{Heat removed} = \frac{\text{material consumption}}{\text{thermal load of the material}}$$

$$= \frac{110 \frac{\text{kg}}{\text{hr}}}{30 \frac{\text{kg}}{\frac{\text{hr}}{\text{ton}}}} = 3.7 \text{ cooling tons}$$

or

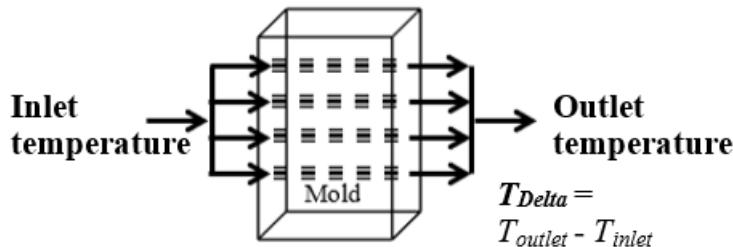
$$= \frac{242 \text{ lbs/hr}}{65 \frac{\text{lbs}}{\frac{\text{hr}}{\text{ton}}}} = 3.7 \text{ cooling tons}$$

According to these calculations the chiller equipment will see a load of 3.7 tons.

Notice that the load obtained in this example is somewhat different from the one calculated in the previous example. They are different methodologies, the first was obtained from the academy and the second from empirical equations developed by the industry. Use the method you prefer, making sure the constants come from reliable sources, such as from your resin manufacturer.

Water flow

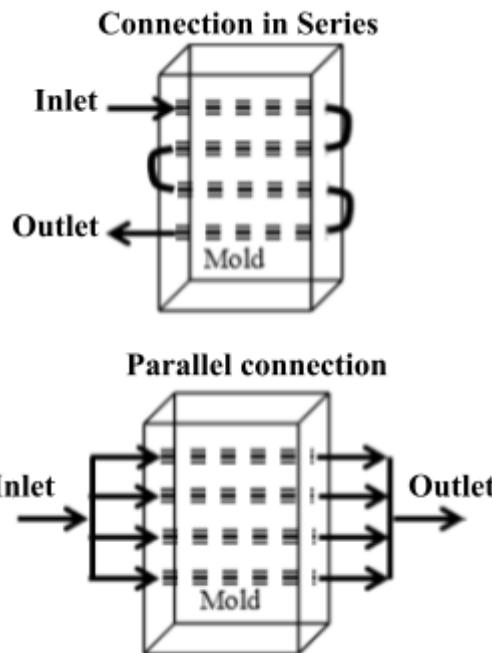
Water flowing through the passages of the mold removes heat from the melted material that is inside the mold. Consequently, the water will observe a rise in temperature; an increase known as Delta T .



V-22. Water flow through the mold and Delta T

The hose connections for cooling the mold should be done in parallel and not in series. A connection in series does not guarantee equal heat removal throughout all mold cavities.

In other words, the cavities at the beginning of the flow will see a heat removal that is distinct from the cavities where the water exits. Even if a connection in series is easier, make the connections parallel.

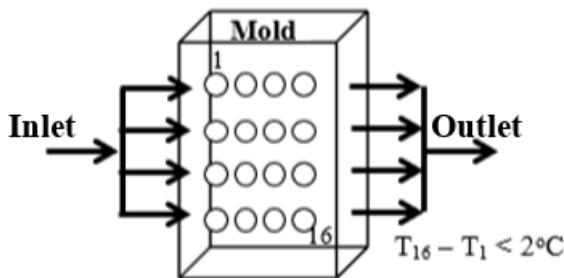


V-23. Water hoses connected to the mold in series and in parallel

A parallel connection will guarantee:

- uniform heat removal and
- uniform temperature distribution in the cavities

The temperature between the cavities should be less than 2°C (4°F).



V-24. Temperature between cavities

Ask the mold designer which is the optimal cavity temperature differential for your mold. To clarify, we mean the temperature of the metal in the cavities. Although water is the one doing the heat removal, the temperatures of the metal in the cavities are the most significant.

Another effect about which you should know is that the water flowing through the mold should be turbulent and not a laminar flow, since turbulent flow is more efficient at removing heat.

Turbulence is achieved by increasing the water flow that enters the mold. The flow rate must be enough to ensure a factor, called the Reynolds number, that indicates whether the flow is turbulent or laminar. This factor is a function of the flow and geometry of the cooling channels.

A simple way to ensure turbulence is by reducing the Delta T of water moving through the mold.

For example:

- Fast processes, less than 10 seconds, Delta $T \sim 1^\circ\text{C}$ (2°F)
- Slow processes, longer than 30 seconds, Delta $T \sim 2^\circ\text{C}$ (4°F)
- Intermediate processes, Delta $T \sim 1.7^\circ\text{C}$ (3°F)

Ask the mold designer for the most suitable water flow, water temperature, and Delta T for your mold.

Water flow is determined with the following equation:

$$\text{Water flow} = \frac{\text{heat removed from the plastic per cycle}}{(\text{specific heat of the water}) * (\Delta T)}$$

The specific heat (C_p) could be that of plain water, extremely hot processes could use oil, and very cold processes could use a mixture of water and antifreeze. Obtain the C_p value that corresponds to the heat transfer fluid that is being used.

The specific heat of water at 13°C (55°F) = **4.196 (kJ/(kg°C))** or **1.003BTU/(lb°F)**

Example: A process consumes 190 grams of polypropylene every 12 seconds. The water temperature entering the mold is 13°C (55°F) and, given that it is an average cycle, we can assume a Delta T of 1.7°C (3°F). What would be the required heat consumption and water flow?

- The energy required for PP is **670 J/gr**, according to the table of energy required for the material.
- The specific heat of water at 13°C (55°F) = **4.196 kJ/(kg °C)**
- Water density = **1kg/liter**

Heat removed =

$$\frac{(\text{total quantity of material}) * (\text{energy required for the material})}{(\text{process cycle time})}$$

$$= \frac{190 \text{ gr} * 670 \text{ J/gr}}{12 \text{ s}}$$

$$= 10608 \text{ J/s} = \mathbf{10.61 \text{ kJ/s}}$$

$$\text{Water flow} = \frac{\text{heat removed from the plastic per cycle}}{(\text{specific heat of the water}) * (\Delta T)}$$

$$= \frac{\frac{10.61 \text{ kJ}}{\text{s}}}{\left(4.196 \frac{\text{kJ}}{\text{kg}^{\circ}\text{C}}\right) * (2^{\circ}\text{C})}$$

$$= 1.264 \text{ kg/s} = 1.264 \text{ liters/s} = 75.85 \text{ liters/min}$$

= 20.1 gpm

This process will require a water flow of 75.9 liters/min or 20.1 gpm.

A simpler way to determine the required water flow of a chiller for a process is with the empirical equation:

$$gmp = \frac{24 * (\text{chiller tons})}{\Delta T}$$

Where:

- gpm = gallons of water per minute
- chiller tons = chiller tons required to cool the mold
- ΔT ($^{\circ}\text{F}$) = outlet water temperature – inlet water temperature

Select the ΔT that best fits your application:

- fast processes, less than 10 seconds, $\Delta T \sim 1^{\circ}\text{C}$ (2°F)
- slow processes, longer than 30 seconds, $\Delta T \sim 2^{\circ}\text{C}$ (4°F)
- average processes, $\Delta T \sim 1.7^{\circ}\text{C}$ (3°F)

$$\Delta T (\text{°F}) = 9/5 \times \Delta T (\text{°C})$$

In the previous example, the process consumes 190 grams of polypropylene every 12 seconds with a ΔT of 1.7°C (3°F). What would be the required heat consumption and water flow?

$$\begin{aligned} \text{Material consumption} &= \\ \text{total quantity of material / process cycle time} \\ 190\text{g}/12\text{s} &= 15.8\text{g/s} = \mathbf{125 \text{ lb/hr}} \end{aligned}$$

According to the “Thermal load for refrigeration” table previously shown, we find that polypropylene needs 35 lb/hr/ton.

The load in tons is obtained with:

$$\text{Heat required} = \frac{\text{material consumption}}{\text{thermal load of the material}}$$

By substituting, we get:

$$\begin{aligned}\text{Heat required} &= (125 \text{ lb/hr}) / (35 \text{ lb/hr/ton}) \\ &= 3.6 \text{ chiller tons}\end{aligned}$$

The required water flow of a chiller for the process is obtained with the equation:

$$\begin{aligned}gmp &= \frac{24 * (\text{chiller tons})}{\Delta T} \\ &= 24 \times (3.6 \text{ chiller tons}) / (3^\circ\text{F}) = 28.8 \text{ gpm}\end{aligned}$$

According to this method the chiller will see a load of 28.8 gpm at a ΔT of 3°F . Again, the result obtained is somewhat different from the previous method since this equation is an empirical approximation and the constants come from different sources.

Estimated Cooling Time

Cooling time is a ***Universal*** control parameter that can affect the dimensional result of molded parts and can also affect the recovery stage. Cooling time, on its own or combined with the temperature of the mold, can affect the thermal dimensions of the product.

An initial cooling time must be determined before performing a process optimization laboratory. The cooling time is initially adjusted with an excessive value and is then optimized with a molding laboratory. This is adjusted higher than required to prevent it from affecting the determination of other parameters that were set before optimizing the cooling time.

This extended cooling time value can be obtained in a simple or a calculated way.

The simple way is to ask the mold manufacturer or someone you trust, who has molded with a similar mold. Take the recommended cooling time and add 30%.

$$\begin{aligned} \text{Extended cooling time} \\ = \text{recommended cooling time} \times 1.3 \end{aligned}$$

The calculated method is determined by using the following equation:

$$E = -\frac{G^2}{2\pi\alpha} \ln \ln \left(\frac{\pi}{4} \frac{(T_x - T_M)}{(T_m - T_M)} \right)$$

T_x = deflection temperature

T_M = mold temperature

T_m = melt temperature

G = part thickness

α = thermal diffusivity

The result of this equation may have up to a 30% error, yet for our purpose it turns out to be a good tool.

Material	α	T_m (°F)	T_M (°F)	T_x (°F)
ABS	0.000185	475	135	203
CA, CAP	0.000181	400	110	192
CAB	0.0002	400	110	201
HIPS	0.000059	440	85	185
IONOM	0.000148	440	85	125
LDPE	0.000176	390	75	113
MDPE	0.000194	340	75	155
HDPE	0.000217	480	75	186
PA 6, 6/6	0.000109	530	150	356
PC	0.000132	560	180	280
PET	0.000138	540	120	153
PP	0.000077	470	105	204
PP/PS	0.000144	530	185	234
PPS	0.000166	630	210	210
PS g.p.	0.000087	420	85	180
PSU	0.000149	700	250	345
PVC	0.000107	380	85	156
PVC rig	0.000123	380	85	174
SAN	0.000088	450	150	225

V-25. Table of constants for cooling time equation

Because the value we are looking for should be larger than is required, we add 40% to the calculated option.

$$\text{Extended cooling time} = E \times 1.4$$

Temperature Control Units (TCU)

A temperature control unit, or TCU, is responsible for maintaining a constant flow and temperature.

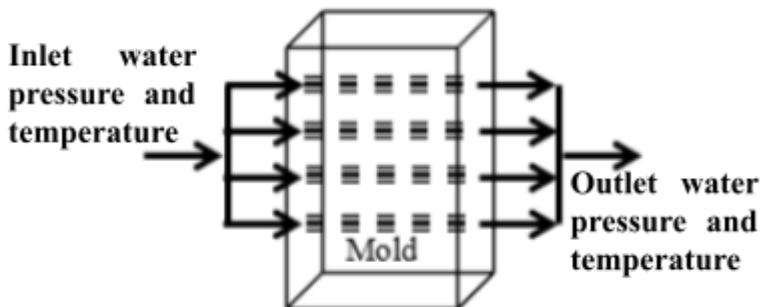
During molding, the removal of heat from the mold is what determines thermal dimensions. Remember, these dimensions are a function of

shrinkage. Water flow, as well as water temperature, are factors that we must control during heat removal.

Other parameters we need to understand are the water's pressure loss (Delta P) and temperature loss (Delta T).

$$\text{Delta } P = \\ \text{inlet water pressure} - \text{outlet water pressure}$$

$$\text{Delta } T = \\ \text{outlet water temperature} - \text{inlet water temperature}$$



V-26. Inlet and outlet water temperature and pressure

During the process, Delta T and Delta P must not change. Any change in these could lead to a variation in the thermal dimensions of the molded parts.

Variation in Delta T could be a result of:

- dirty channels
- changes in the process
- changes in water flow
- change in melt temperature

Variation in Delta P could be a result of:

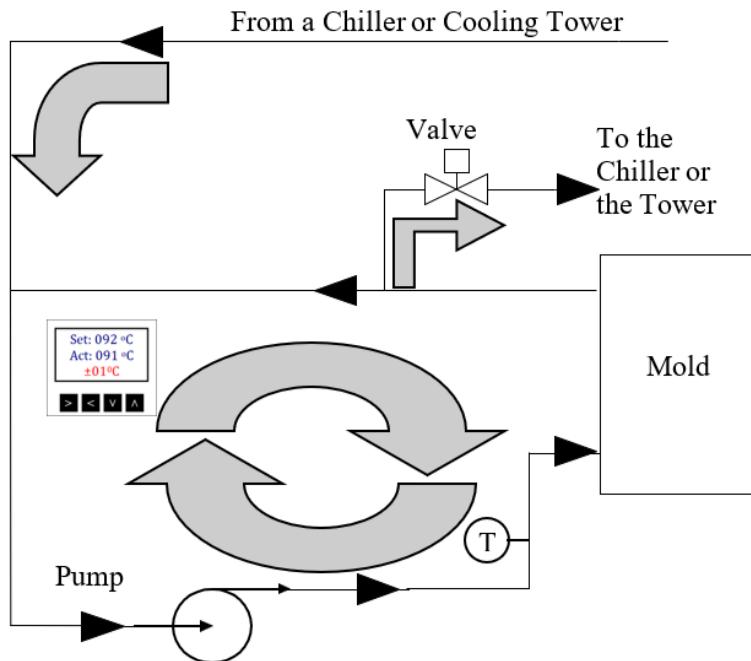
- dirty channels
- channels blocked by an object
- incorrect connection
- piping of a smaller diameter

TCU with direct cooling

This uses a pump and a valve to maintain a constant temperature and water flow. The source of cooling comes from either a chiller or a cooling tower.

The main components are:

- pump: responsible for guaranteeing the flow of water through the mold.
- cold water intake: allows cold water from the chiller or cooling tower to enter the TCU.
- cooling valve: discharges hot water from the TCU to the chiller or the tower.
- thermocouple: measures the water temperature entering the mold.
- control: the brain of the TCU, which controls parameters such as the temperature of water going to the mold.

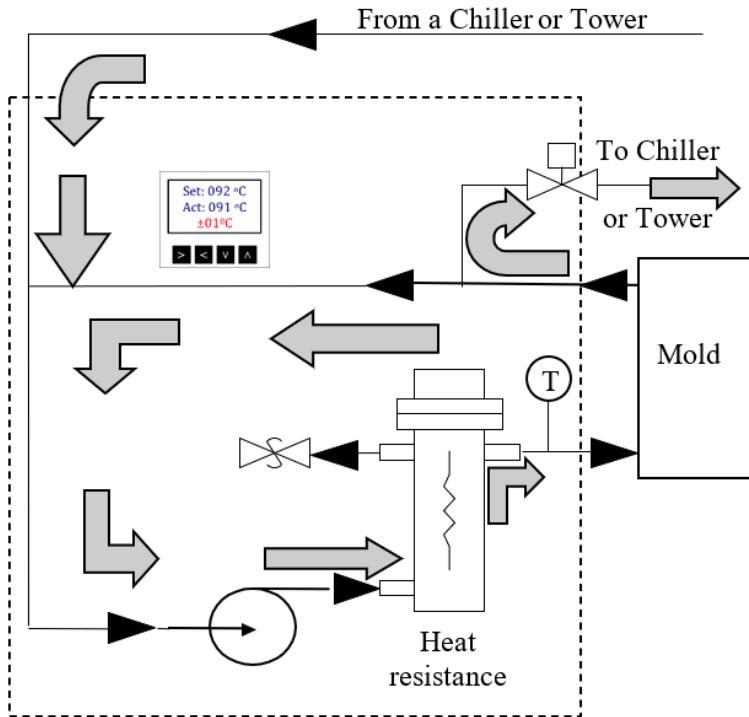


V-27. Diagram of direct cooling components

TCU with direct cooling and heating

This unit is similar to the direct cooling TCU. A constant water flow and constant water temperature are maintained using a pump, a valve, and a heater. The cooling source comes from a chiller or cooling tower, and the heater helps raise the temperature.

The biggest difference is the heater. The heater only activates to effectively increase the water temperature.



V-28. Diagram of direct cooling and heating components

Let's look at this operation in more detail:

The pump is responsible for guaranteeing the water flow between the TCU and the mold. When the control detects that the water's temperature to the mold is increasing it opens the valve, allowing that hot water to return to the chiller or the tower. Cool water then enters the TCU from the suction side of the pump. The heater only activates to increase the water temperature when the process requires it.

From an energy cost point of view, adding a heater appears to be inefficient, but that is not necessarily so. The heater only activates when needed. Let's say that during a mold startup it is necessary to increase 10°C; it would take too much time to reach that operating temperature if we depended on the friction and heat of the mold. Another example, during a thermal dimension optimization lab, where various water temperatures must be evaluated to study an effect, we would have to wait too long between each temperature increment.

There are molds that need to be connected to more than one TCU. This could be to improve the demolding of the parts, to correct bending, to guarantee some mechanical characteristic of the molded part such as stopping the formation of crystals in a specific location of a molded product, etc.

There are other styles of TCUs:

- *negative flow* - Using suction, negative water pressure is achieved in the mold, solving filtration problems.
- *closed circuit* - This separates the mold water from the process water, in order to minimize deposits of contaminants and minerals in the mold.
- *with oil* - The transfer fluid is oil, which permits high temperature ranges, from 175°C to 290°C (~350°F to 550°F).
- *integrated chiller and TCU* – This does not require an external cooling source.

Each TCU provider offers multiple options. Consider:

- Mold purging, a device that allows you to empty the water from the mold, avoiding water spillage during mold change or maintenance.
- Protective devices, such as programmable pressure and temperature alarms.
- Alarms preset by the manufacturer, such as the system's maximum operating temperature and minimum pressure allowed from its cooling source (chiller or tower). These alarms mainly protect your equipment.

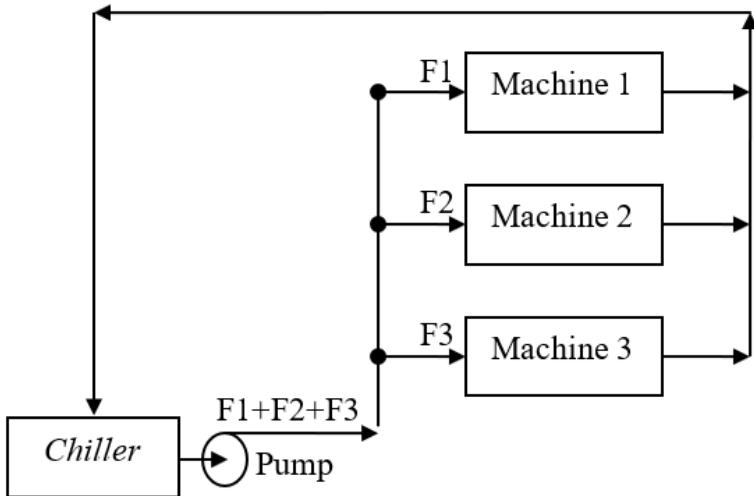
It is important not to disable those alarms included by the manufacturer. For example, the minimum water pressure alarm coming from the cooling source, typically preset above 1 bar, prevents the pump from being damaged by cavitation.

It is also important that the temperature of the cooling source (chiller or tower) must be at least 5°C (10°F) cooler than the temperature that you want to set the TCU. This is done to compensate for heat loss due to friction and heat coming from the mold; otherwise, it may not be able to drop to the set temperature.

If the cooling tower provides water at 30°C, connect to TCUs that operate at 35°C or more. Any mold that requires less than 35°C must be connected to a chiller.

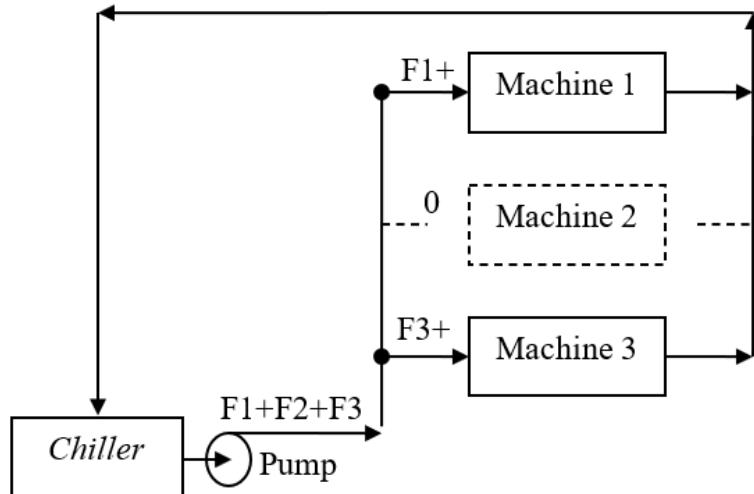
A significant benefit of TCUs is that regardless of what happens in the rest of the factory, the pump ensures a constant flow between the TCU and the mold. In other words, if the TCU did not exist and the cooling came directly from a centralized chiller, or was being shared with other processes, the mold would be subject to anything that was happening with that shared water pipe.

For example, consider three machines with similar molds that share a chiller. Under an ideal operation the chiller water flow is shared between the three machines.



V-29. Three machines sharing a chiller

If one of the machines is stopped for mold maintenance, the flow of chilled water that previously reached three molds will be shared by only two molds.



V-30. Three machines sharing a chiller, with a stopped machine

This could consequently affect the thermal dimensions of the molded parts. In the same way, imagine that on a Monday, all the machines are

started up after a weekend shutdown. Each time a machine is powered up, the ones that are already in operation will be affected, a pretty complicated scenario.

Summary:

- Some of the heat transmitted to the water, circulating between the TCU and the mold, comes from friction.
- Hose connections must be in parallel.
- Water flow should be turbulent.
- The Delta T of the water must be less than 2°C (4°F).
- The metal temperature of the cavities must be uniform, with a differential of less than 2°C (4°F).
- Pressure loss (Delta P) through the mold should be maintained.
- Temperature loss (Delta T) through the mold should be maintained.
- In centralized systems the TCU independently isolates the flow of water to the mold.
- The cooling source (chiller or tower) must be at least 5°C (10°F) cooler than the TCU's set temperature.

Questions

- 1) A mold requires water at a temperature of 32°C, a chiller provides water at 12°C (~55°F) and a tower at 30°F (~85°F). Where should the TCU be connected?
 - a. to a tower, because tower water must be at least 1°C colder than the desired temperature
 - b. to a chiller since tower water is only 2°C colder and is not 27°C (32°C -5°C) as required
- 2) For PA (nylon), how many cooling tons are needed for a consumption of 40 lb/hr?
 - a. 2 cooling tons
 - b. 1 cooling ton
 - c. 1/2 cooling ton
- 3) A mold that creates nylon components consumes 275 grams per cycle, and the expected cycle is 9 seconds. How much is the consumption per hour?
 - a. Material consumption = 275 g/hr
 - b. Material consumption = (275/9) - 39 lb/hr
 - c. Material consumption = $(275\text{g}/9\text{s}) \times (3600 \text{ s/hr}) \times (\text{kg}/1000\text{g}) = 110 \text{ kg/hr}$
- 4) The connections between the manifold and the mold are made
 - a. in parallel, and the hoses are the same diameter as the mold connections.
 - b. in series, and the hoses are a smaller diameter than the mold connections.
- 5) The temperature of the mold is equal to the water temperature.
 - a. True, the temperature of the metal is somewhat cooler.
 - b. False, the water and metal temperatures could be different.
- 6) A process using polypropylene (PP) needs 3.6 tons of chilled water at a Delta T of 2°C (4°F). What chiller water flow is needed?
 - a. $24 \times (3.6 \text{ chiller tons}) / (4^\circ\text{F})$
 - b. $30 \times (3.6 \text{ chiller tons}) / (4^\circ\text{F})$
 - c. $24 \times (3.6 \text{ chiller tons}) / (2^\circ\text{C})$

VI. Molding from the Desk

- **Press Calculations**
- **Injection Unit Calculations**

Before proceeding with a molding lab, you should have determined several factors such as:

- clamping force
- space between tie-bars in the platen
- maximum and minimum opening of the mold
- nozzle tip and bushing
- transfer position to hold
- barrel temperature
- injection unit size
- backpressure
- recovery position
- intensification ratio

These initial calculations (Molding from the Desk) give you a starting point and save you time during startup. Avoid costly errors when determining these initial parameters.

Press Calculations

Your goal should be to identify your process' needs and address those needs with well-thought-out solutions. Because of this:

- Before molding, or making some adjustments to the injection machine, you should make some initial calculations.
- We call these initial calculations "Molding from the Desk".
- Remember that you are working with expensive equipment; do not rush the job.

This section will cover:

- clamping force fundamentals
- projected area
- thin wall calculation
- forces resulting from side action mechanisms
- three-plate molds
- stack molds
- platen spacing on machines with tie bars
- platen spacing on tie-bar-less machines
- minimum and maximum press openings
- ejector patterns

Clamping force fundamentals

There is a difference between the clamping force the press is capable of, and the force required to keep the mold closed. The melt that flows into the mold cavities enters at high pressure, and the press must generate the force necessary to contain that pressure.

Force is usually measured in US tons (2000 lbf) or metric tons in kilo-Newtons (kN).

$$\text{US ton} = \mathbf{8.90 \text{ kilo-Newton (kN)}}$$

$$\text{metric ton} = \text{US ton} \times 1.10 = \mathbf{9.81 \text{ kilo-Newton (kN)}}$$

We can determine the clamping force with the following equation:

$$\text{Force} = \text{pressure} \times \text{area}$$

Melt pressure varies with the type of material. For example, each material has a pressure factor measured in units of force/area.

Some of these pressure factors are:

Material	US ton/in ²	kN/cm ²	
Polypropylene	1.5	3.5	2.1
High density polyethylene	1.5	2.5	2.1
Low density polyethylene	1.0	2.0	1.4
Nylon 66	3.0	5.0	4.1
Polycarbonate	3.0	5.0	4.1
Flexible PVC	1.5	2.5	2.1
Rigid PVC	2.0	3.0	2.8
Polystyrene	2.0	4.0	2.8

VI-1. Table of pressure factors for some materials

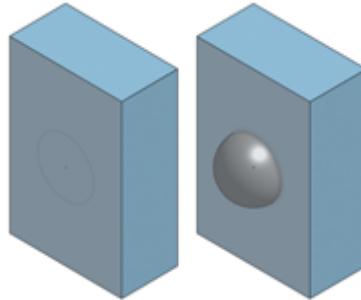
The table is a reference only; corroborate each resin's specifications with its manufacturer.

Projected area

The projected area is the plane or surface that you would see in the mold's partition, or parting line. For example:

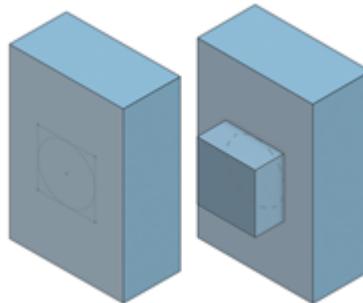
- A sphere's parting line would look like a circular plane, and the area would be equal to:

$$\text{diameter}^2 \times 3.1416 / 4$$



VI-2. Projected area of a sphere

- A cube's parting line would look like a square or rectangular plane. The area would be the length multiplied by width.



VI-3. Projected area of a cube

- In a cup, the largest diameter is at the mold partition. The area would be equal to:

$$\text{largest diameter}^2 \times 3.1416 / 4$$



VI-4. Projected area of a cup

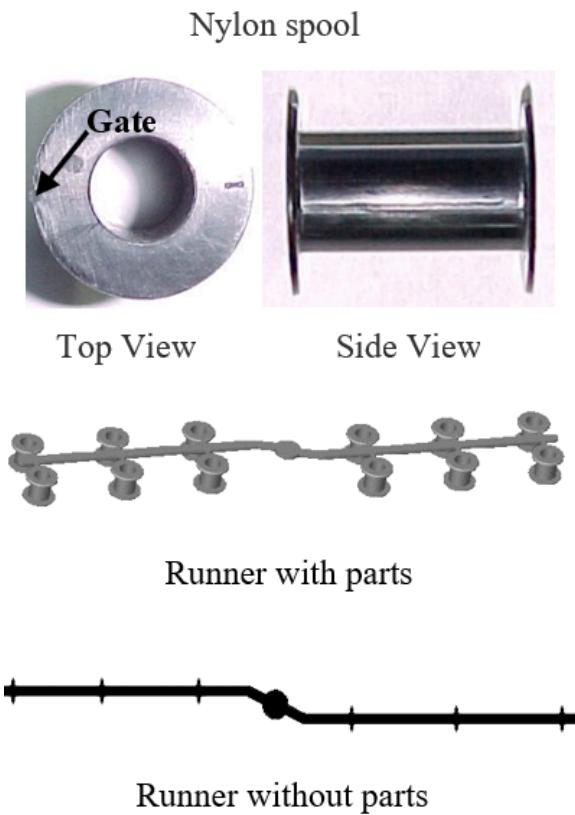
It's simple; if the projected area increases, the required clamping force will also increase. During the determination of a projected area, ignore the depth of the cavity.

There are several aspects to consider when determining the clamping force:

- the material and its characteristics
- understanding how the melt flow fills the cavities
- projected area of all cavities
- projected area of the runner
- type of mold, three plates or stack mold
- springs in the mold partition that act against the clamping force
- sliders with side action that add load to the clamping force
- that the clamping force of the press should always be greater than the clamping force required by the mold

Do not set the clamping force to maximum; excessive force could damage the cavities over time, especially the vents. Remember that gases inside the cavities and runner are expelled through small openings during fill. These very small spaces, or vents, allow these gases to escape. If the clamping force is excessive, the vents could choke. When this happens, it makes it difficult for the gases to escape. Some of these gases come from the melt, and at high pressures these gases could combust, an action known as "dieseling". Molded parts will show a burn at the end of the fill, near where the choked vents are.

Example (courtesy of Peter Paul Electric):
Nylon spools are molded using a 12-cavity mold.



VI-5. Nylon spools and runners with and without parts

In this example the clamping force would be the result of:

- projected area of all cavities
- projected area of the runner
- slider action

The projected area corresponds to 12 circles with a hole in each center plus the area of the runner. The projected area of a circle with a hole in the center is calculated by subtracting the area of the minor (interior) diameter from the area of the major (exterior) diameter:

$$\text{Area of the parts} = \frac{D_{\text{exterior}}^2 - D_{\text{interior}}^2}{4} \pi$$



$$\text{Area} = (2.21^2 - 1.07^2) \times 3.1416 / 4 = \mathbf{2.94 \text{ cm}^2 (0.46 \text{ in}^2)}$$

Projected area of the runner:



VI-6. Projected area of one of the 12 spools and its runner

The projected area of the runner can be simplified with a simple approximation using rectangles and a circle. Consider half of the runner composed of two rectangles (with area of length times width) and a circle (with area of diameter² x π/4).

The area of the runner would be the sum of four rectangles and a circle.



VI-7. Area of the runner

$$\text{Projected area of the runner} = 35.29 \text{ cm}^2 (5.47 \text{ in}^2)$$

Note that, for this approximation, the area where the gates are is not considered; we assume that amount is insignificant.

With irregular geometry, where the calculation of the area cannot be performed by conventional equations, graph paper is a good option. Trace or draw the component on the paper, counting the squares within the drawing (considering 1/2 and 1/3 squares if possible), then multiplying by the area of each square.

Another alternative would be to use the area function of a CAD program; for this you will need a drawing of the component in digital form.

Finally, add all the areas:

$$\begin{aligned}\text{Total area} &= \text{area of the runner} + (12 \times \text{spool area}) \\ &= 35.29 + (12 \times 2.94) = \mathbf{71 \text{ cm}^2} \\ &= 5.47 + (12 \times 0.46) = \mathbf{11 \text{ in}^2}\end{aligned}$$

From resin manufacturer XYZ, a clamping force factor was obtained for nylon of 4.1 to 6.9 kN/cm² (3 to 5 USton/in²).

If we use a pressure factor of 4.1 kN/cm² (3 USton/in²), the required force is:

$$\begin{aligned}71 \text{ cm}^2 \times 4.1 \text{ kN/cm}^2 &= \mathbf{291 \text{ kN}} \\ (11 \text{ in}^2 \times 3 \text{ ton/in}^2) &= \mathbf{33 \text{ USton}}\end{aligned}$$

If we use a pressure factor of 6.9 kN/cm² (5 USton/in²), the required force is:

$$\begin{aligned}71 \text{ cm}^2 \times 6.9 \text{ kN/cm}^2 &= \mathbf{490 \text{ kN}} \\ (11 \text{ in}^2 \times 5 \text{ ton/in}^2) &= \mathbf{55 \text{ USton}}\end{aligned}$$

You should be wondering; What pressure factor do we use, 4.1 or 6.9 kN/cm²? The pressure factor will depend on the difficulty of filling the mold.

For example, if we consider that:

- thin walls will require higher fill pressures than thick walls
- a large fill travel will require more pressure than a short fill travel

The pressure factor will depend on a thin wall calculation, a value that considers the fill distance and the thickness of the spaces.

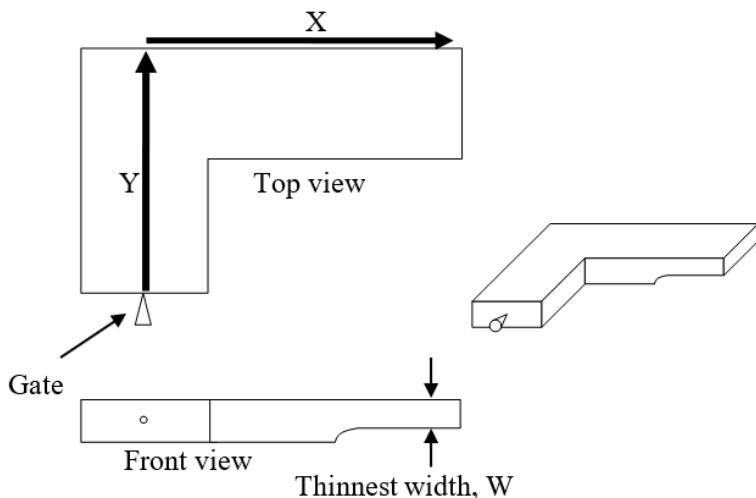
Thin wall calculation

A thin wall calculation is a factor that represents fill difficulty. This factor takes into consideration the distance that the melt must travel and how narrow those passages are. This factor is represented by the following equation:

$$\text{Thin wall calculation} = \frac{\text{farthest flow path}}{\text{thinnest wall on the path}}$$

- Thin wall calculation: a value that represents the fill difficulty; this difficulty increases when the value increases
- farthest flow path: the path of the flow from the gate to the farthest fill point
- thinnest wall on the path: the width of the thinnest wall on the farthest flow path

In the example below, there is an L-shaped part.

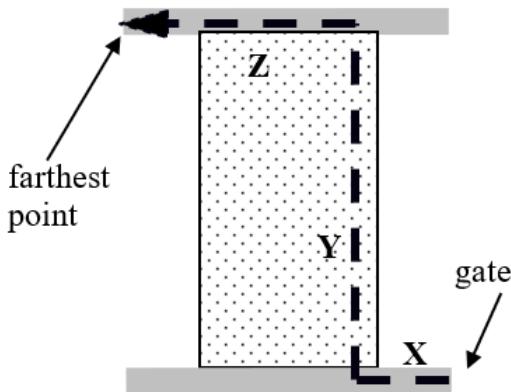


VI-8. Example of an L-shaped part

The farthest flow path can be approximated by the sum of X and Y . The thinnest wall of the path is W . So:

$$\text{Thin wall calculation} = (X + Y) / W$$

Let's continue with the spool's thin wall calculation. The path from the gate to the selected farthest point is indicated by the dashed line.



VI-9. Flow path of a spool

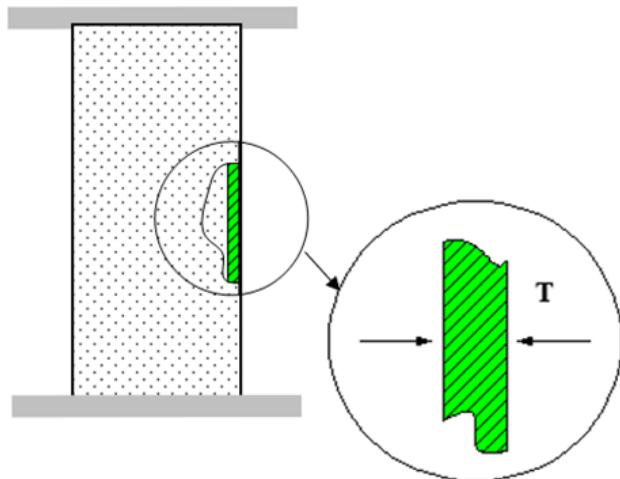
The farthest flow path would be the sum of X , Y , and Z .

You must be wondering, if the spool is a cylindrical part with a hole in the center, how can the path be represented linearly?

Good question, the melt will flow along the path of least restriction, and the path of Y is probably diagonal and around the circumference of the spool. If you have a numerical flow analysis program, excellent; use it. However, remember that we are molding from the desk; do not try to complicate your life with calculations that will not necessarily give you the best results.

$$\text{Farthest flow path} = X + Y + Z = \mathbf{50 \text{ mm (1.97 in)}}$$

Now let's look at the thinnest wall from the path chosen in the illustrated drawing, where we find that the thinnest wall was found to be the central wall of the spool; and let's call this thickness T .



VI-10. Thickness (T)

$$T = 1.22 \text{ mm (0.048 in)}$$

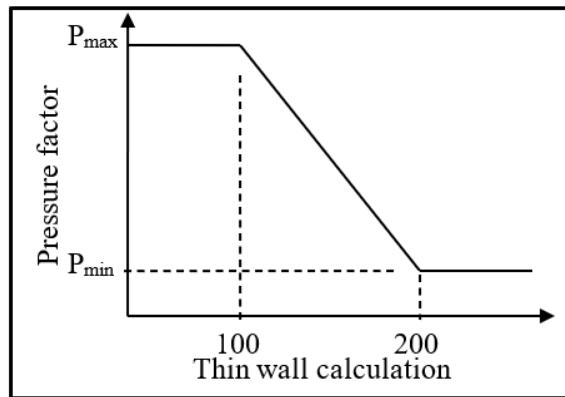
Select the thickness within the chosen path. Remember that there is a relationship between the flow path and the thickness of the walls. If you want, you can do several thin wall calculations with distinct paths. Once again select the thinnest wall within each chosen path; then use the largest value calculated as it will represent the worst condition.

$$\begin{aligned} \text{Thin wall calculation (TW)} &= 50 \text{ mm / } 1.22 \text{ mm} \\ &= 41 \end{aligned}$$

TW	Criteria
≥ 200	Use the highest pressure factor. Force = (projected area) x (highest pressure factor)
≤ 100	Use the smallest pressure factor. Force = (projected area) x (smallest pressure factor)
between 100 and 200	Interpolate between pressure factors. Force = (projected area) x (interpolated pressure factor)

VI-11. Table of thin wall criteria

The interpolated pressure factor is obtained with a linear interpolation.



VI-12. Linear interpolation of the pressure factor

$$\text{Pressure factor} = \frac{(TW - 100) * (P_{max} - P_{min})}{100} + P_{min}$$

With the spool example, we have a nylon pressure factor of 4.1 to 6.9 kN/cm² (3 to 5 USton/in²) and a thin wall value of 41. According to the indicated criteria, if less than 100, we will use the minimum pressure factor, 4.1 kN/cm² (3 USton/in²).

That means that the normal clamping force, as a result of the projected area and the pressure factor, is:

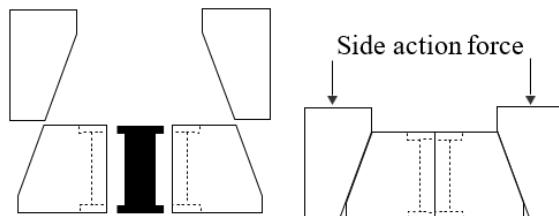
$$\begin{aligned}\text{Normal force} &= \text{pressure} \times \text{area} \\ &= 71 \text{ cm}^2 \times 4.1 \text{ kN/cm}^2 = \mathbf{291 \text{ kN}} \\ &(\mathbf{=11 \text{ in}^2 \times 3 \text{ ton/in}^2 = 33 \text{ USton}})\end{aligned}$$

Forces resulting from side action mechanisms

One of the reasons for selecting the spool as an example is because it includes more than a simple force calculation due to the projected area in the partition of the mold. The clamping force can also be affected by forces resulting from mechanisms that act laterally.

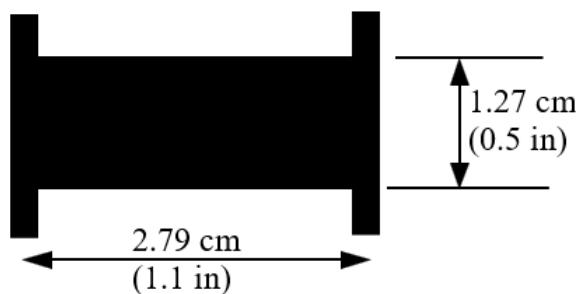
These mechanisms are part of the mold cavities and stay in position while the mold is closed. After the mold opens those mechanisms move, freeing the molded parts. These lateral forces are reflected in the clamping force, since it is the press that keeps these mechanisms in position.

Notice in the illustration how the cavity separates to free the molded part.



VI-13. Side action mechanisms (sliders) driven by clamping force

Each cavity is split in the middle by means of two sliders that will experience a force as a result of the melt pressure. This force will be the result of melt pressure (4.1 kN/cm^2) multiplied by the projected area in the cavity partition. With the area in the partition, we can assume that the extremities have an insignificant area, so our calculations will be based on the area of a rectangle.



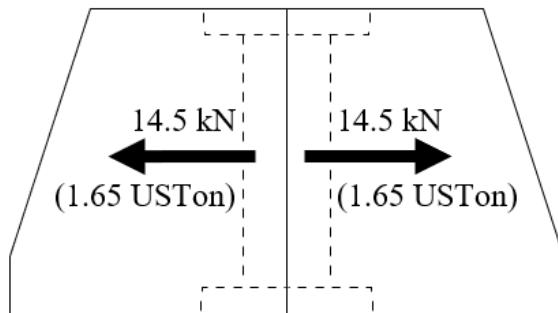
VI-14. Measurements to calculate the projected area

$$\begin{aligned} \text{Lateral projected area} &= \\ 1.27 \text{ cm} \times 2.79 \text{ cm} &= 3.54 \text{ cm}^2 \\ (0.5 \text{ in} \times 1.1 \text{ in}) &= 0.55 \text{ in}^2 \end{aligned}$$

Then:

$$\begin{aligned}\text{Lateral force per slider} &= \\ \text{lateral projected area} \times \text{pressure factor} &= \end{aligned}$$

$$\begin{aligned}3.54 \text{ mm}^2 \times 4.1 \text{ kN/cm}^2 &= \mathbf{14.5 \text{ kN / slider}} \\ (0.55 \text{ in}^2 \times 3 \text{ USton/in}^2) &= \mathbf{1.65 \text{ USton / slider}}\end{aligned}$$



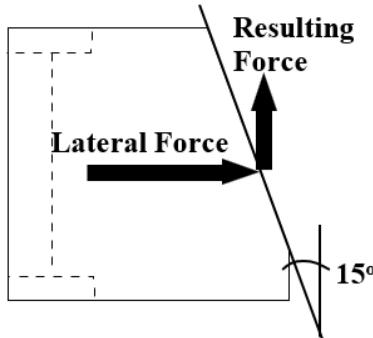
VI-15. Lateral force per slider

According to this number, each half of the cavity will see a force of 14.5 kN (1.65 USton) trying to move the sliders away.

Considering that there are 12 cavities and that each is kept closed by wedges, the total lateral effect could be determined by the following equation:

$$\begin{aligned}\text{Lateral force} &= \\ \text{force per slider} \times 2 \text{ sliders} \times 12 \text{ cavities} &= \\ = 14.5 \text{ kN} \times 2 \times 12 &= \mathbf{348 \text{ kN}} \\ (1.65 \text{ USton} \times 2 \times 12) &= \mathbf{40 \text{ USton}}\end{aligned}$$

This number, 348 kN (40 USton), is the total force that the melt will exert laterally against the wedges.



VI-16. Resulting force

These wedges are manufactured with an angle, in this case 15°, and only a fraction of this lateral force will be seen in the direction of the closing of the wedges.

This resulting force in the direction of the press is determined by multiplying the lateral force by the tangent of the wedge's angle:

$$\begin{aligned}
 &\text{Force as a result of lateral action} \\
 &= \text{lateral force} \times \tan(15^\circ) \\
 &= 348 \text{ kN} \times 0.27 \text{ (40 USton} \times 0.27\text{)} \\
 &= \mathbf{94 \text{ kN (10.8 ton)}}
 \end{aligned}$$

The total clamping force required for the mold would be the sum of the force resulting from the melt in the mold partition and the lateral action force of the cavities.

$$\begin{aligned}
 \text{Final force} &= \text{normal force} + \text{lateral action force} \\
 291 \text{ kN} + 94 \text{ kN} &= \mathbf{385 \text{ kN}} \\
 (33 \text{ USton} + 10.8 \text{ USton}) &= \mathbf{43.8 \text{ USton}}
 \end{aligned}$$

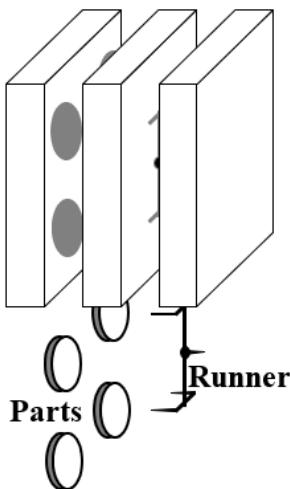
“Molding From the Desk” is an exercise that we must all perform before attempting to adjust the press; even if the mold maker recommends a clamping force, it should be corroborated.

The easiest, but not correct, thing to do would be to adjust the clamping force of the press to its maximum. Excessive force would eventually damage the mold by, for example, deforming the vents in the cavities. If

the vents are blocked, then gases (air plus vapor from the same melt) inside the cavities could experience the dieseling effect, an explosion due to a combustion of gas at high pressures. The next time you find burn marks near the vents on the molded parts, verify the clamping force before attempting to repair the mold.

Other factors to consider are the springs. If your mold uses springs at the partition line and they are compressed when the mold is closed, add the force of each spring. The spring force factor can be obtained from the spring manufacturer.

Three-plate molds



VI-17. Three-plate mold

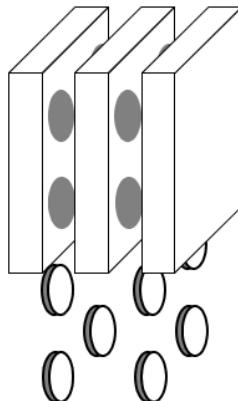
This type of mold molds the parts and the runners in two distinct partitions. The mold is split into three, the parts are molded in one partition and the runner is formed in another. The purpose of this type of design is for molded parts, such as round parts, which need to be filled from the center.

Calculating the clamping force of a three-plate mold must be done twice, in the partition where the parts are molded and the partition where the runner is formed. Then, the largest of the two is selected. It is common for the force of the molded parts to be more than the force of the runner,

but it is not true in 100% of the cases. Do the calculations and select the largest force.

Stack molds

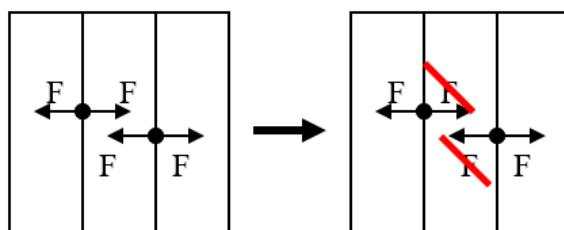
A stack mold has the same number of cavities in each partition side of the mold.



VI-18. Stack mold

If the same number of identical parts come out of each side, the clamping force on each side will be equal. Because of this, we only consider the clamping force of one side.

You must be wondering, why consider only one half instead of using both? That is what makes the stack design attractive, it produces twice the parts without needing twice the clamping force. Look at the illustration showing the force vectors.



VI-19. Stack mold force vectors

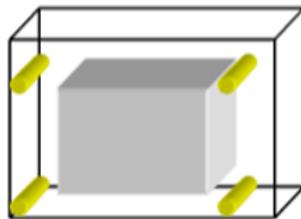
Forces of equal magnitude in opposite directions will try to open each partition of the mold. The center vectors will cancel each other, resulting in the equivalent force of only one side.

In addition to the clamping force, you should verify that the mold fits properly in the press and that the ejector pattern of the press matches that of the mold.

Platen spacing on machines with tie bars

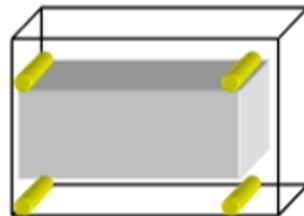
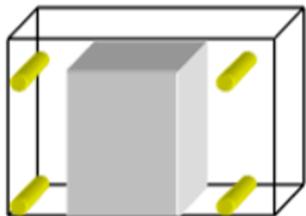
If the press has tie bars, check that the mold fits in between them. It may fit inside the four tie bars, vertically between the bars, or horizontally between the bars.

Between tie bars



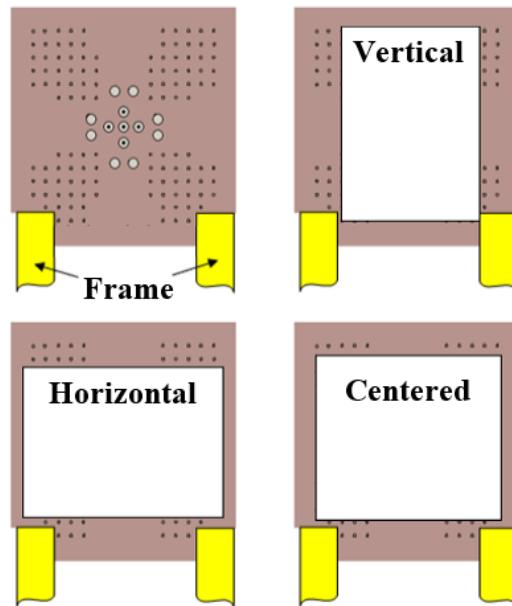
Vertical

Horizontal



VI-20. Platen spaces with tie bars

Platen spacing in tie-bar-less machines

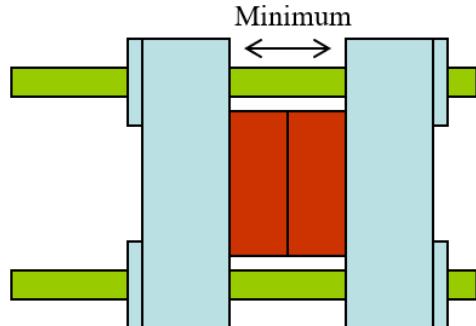


VI-21. Platen spaces on tie-bar-less machines

Tie-bar-less presses provide greater flexibility to accommodate the mold; even so, you must verify the available space.

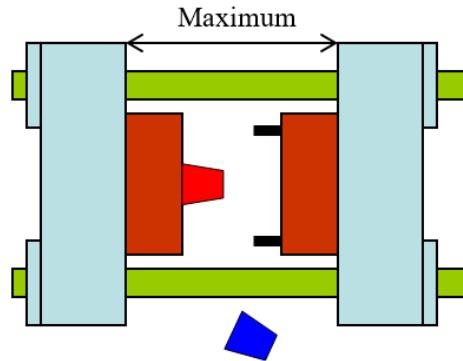
Minimum and maximum openings

Verify that the minimum press opening is enough to press the mold closed.



VI-22. Minimum press opening

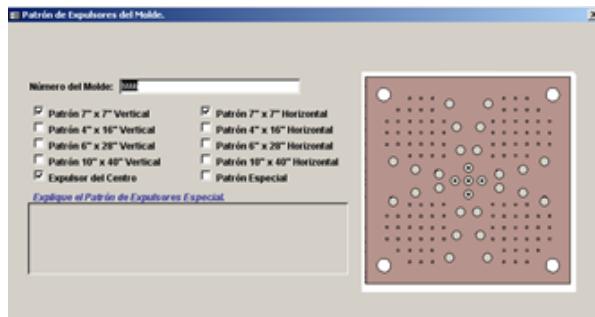
Also, verify that the mold opens enough so that the parts can be demolded.



VI-23. Maximum press opening

Ejector patterns

Verify that your machine has the ejector pattern required for the mold.



VI-24. Ejector pattern

Although most of these verifications appear to be simple and obvious, do your job and verify anyway. Those who have been present when a mold was transferred to the wrong press understand this advice, and those who have not witnessed it need to make the calculations to avoid it.

Questions

- 1) The chosen clamping force must be
 - a. the maximum force that the press is capable of.
 - b. the clamping force required by the mold.
 - c. the maximum capacity of the hydraulic pump.
- 2) There is a 16-cavity mold that makes discs with a diameter of 0.5 inches. The projected area of each disc is 0.2 in^2 . The projected area of the runner is 1 in^2 . The thin wall calculation (TW) gives a value of 50. The material is polystyrene with a pressure factor of 1.5 to 3 ton/in 2 .
 - 2a) What is the total projected area?
 - a. $16 \text{ cavities} \times 0.2 = 3.2 \text{ in}^2$
 - b. $16 \text{ cavities} \times 0.2 + 1 \text{ from the runner} = 4.2 \text{ in}^2$
 - c. 0.2 in^2
 - d. 1 in^2
 - 2b) According to the thin wall calculation ($TW = 50$), the factor for polystyrene is
 - a. 1.5 ton/in^2 since the TW is less than 100.
 - b. 3 ton/in^2 since the TW is greater than 200.
 - c. 2.5 ton/in^2 since the TW is between 100 and 200.
 - 2c) For a factor of 1.5 ton/in^2 , the calculated clamping force is
 - a. $1.5 \times 4.2 \text{ in}^2 = 6.3 \text{ tons.}$
 - b. $3 \times 4.2 \text{ in}^2 = 12.6 \text{ tons.}$
 - c. 1.5 tons.
- 3) Excessive clamping force strangles a mold's vents. This can cause internal combustion since the gases do not find a place to escape. This is resolved by
 - a. increasing the clamping force.
 - b. enlarging the vents.
 - c. cleaning the vents and increasing the clamping force.
 - d. adjusting the press to the required clamping force and cleaning the vents.

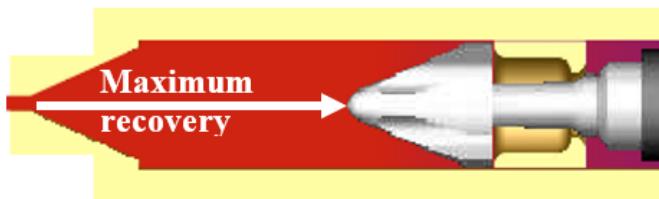
Injection Unit Calculations

In this section we will discuss:

- injection unit size
- nozzle tips and sprue bushings
- fountain flow effect
- density and specific density
- injection speed and injection flow
- barrel utilization
- recovery position
- residence time
- transfer position
- temperature profiles
- intensification ratio
- machine labeling

Injection unit size

The volume of the injection unit represents the maximum material capacity that the injection unit can charge during recovery.



VI-25. Maximum recovery

It is important to know that the recovery amount that the unit is capable of is greater than the amount required by the mold; in general, we say:

Maximum recovery > 30% more than what the mold requires

You may wonder why 30% more than what the mold requires. There are many reasons, such as: to guarantee a cushion after the hold stage, to compensate for the compressibility of the thermoplastic melt, to compensate for melt that returns to the other side of the check ring in the injection barrel, etc.

The factors that determine how much larger the barrel should be include:

- available recovery time
- whether the material is easy or difficult to melt
- whether the material degrades or resists long residence times in the barrel

The capacity of the injection unit is measured in volume, not by mass. If the size of the injection unit comes in mass units (ex: oz or g), you should know that it is generally based on the density of polystyrene ($\rho = 0.94 \text{ g/cc}$ or 0.54 oz/in^3).

Notes:

- Use volume values, not mass values, when specifying an injection unit.
- During your calculations, try to reach the specific density of the melt that is exposed to recovery temperature and pressure, not to ambient temperature and pressure.

Example:

Density of a polycarbonate, PC
= 0.95 g/cm^3 (melt)
= 1.2 g/cm^3 (solid)
= 1.04 g/cm^3 (plasticized)

Use the plasticized density, the density of the melt exposed to recovery conditions. Remember that thermoplastic melt is compressible. Some machine manufacturers call this recovery density the "discharge factor".

Example:

A polycarbonate with a plasticized density of 1.04 gr/cm^3 will fill a mold with a total weight, parts and runner, of 125 gr. According to the material manufacturer, barrel utilization should be between 45% and 60%. What would be the volumetric specifications of the barrel?

$$\begin{aligned}\text{Injection volume required by the mold} &= \\ &\quad \text{weight/density} \\ &= (125 \text{ g}) / (1.04 \text{ g/cm}^3) = \mathbf{120.19 \text{ cm}^3}\end{aligned}$$

$$\text{Utilization volume at 45\%} = 120.19 \text{ cm}^3 / 0.45$$

$$= 267.09 \text{ cm}^3$$

The barrel should be less than 267.09 cm³.

$$\begin{aligned}\text{Utilization volume at 60\%} &= 120.19 \text{ cm}^3 / 0.6 \\ &= 200.32 \text{ cm}^3\end{aligned}$$

The barrel should be more than 200.32 cm³.

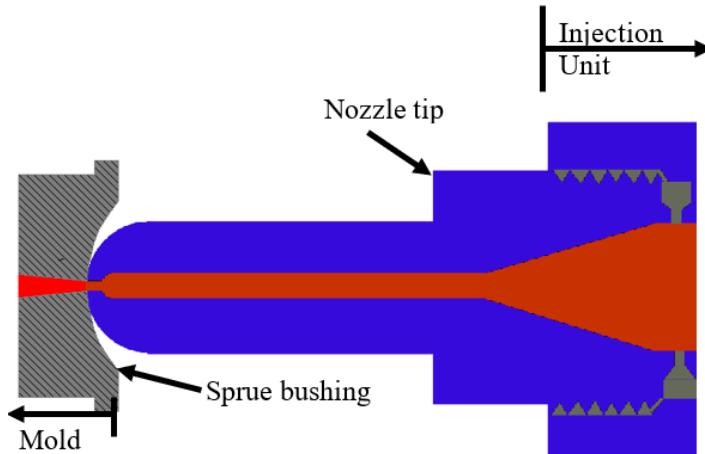
So, to meet the requirements of that PC, the volume of the injection unit should be between 200.32 cm³ and 267.09 cm³.

It is possible that you will not find the density of the material under the recovery conditions and will be forced to use a generic density or use density at ambient temperature and pressure.

Remember that you are working with thermoplastics, materials with properties that are functions of time, temperature, and pressure. If we add to this the large number of injection unit types, we can conclude that some calculation error will be present even with adequate density.

Nozzle tips and sprue bushings

The nozzle tip is installed in the injection unit, and the sprue bushing is installed in the mold. The injection unit generates the force required to keep them coupled together. If they separate while the mold fills, the melt could filter between them. As a rule, think of the nozzle tip as part of the mold.



VI-26. The nozzle tip and sprue bushing

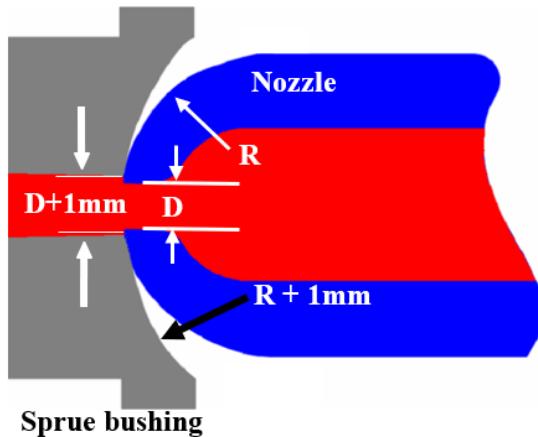
The nozzle tip is an important component, because of this the proper one should be selected and maintained at a specific temperature. This has more significant effects with semi-crystalline materials than with amorphous materials.

For example, with an amorphous material X, if the temperature is 50°F above the required value, the nozzle could create a drooling effect and strings might be seen during the expulsion of the sprue. If the temperature is 50°F under the required value, the nozzle could get stopped up or create cold slugs that could migrate to the mold.

Now, with a semi-crystalline material Y, if the temperature is only 5°F over the required value, the nozzle could create the drooling effect, and strings might be seen during the expulsion of the sprue. If the temperature is only 5°F under the required value, the nozzle could get plugged or create cold slugs that could migrate to the mold.

Diameter of holes and contact radii

Let's look at the diameter D and radius R of the nozzle tip in the illustration.

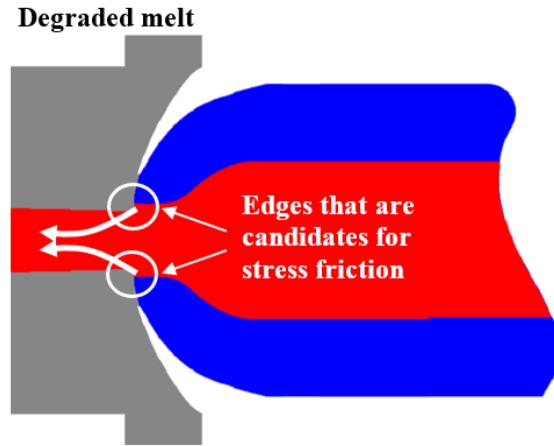


VI-27. Diameters and radii of the nozzle tip and sprue bushing

The diameter of the bushing's opening is 1 mm greater than diameter D of the opening of the nozzle tip. The contact radius of the bushing is 1 mm greater than radius R at the nozzle tip.

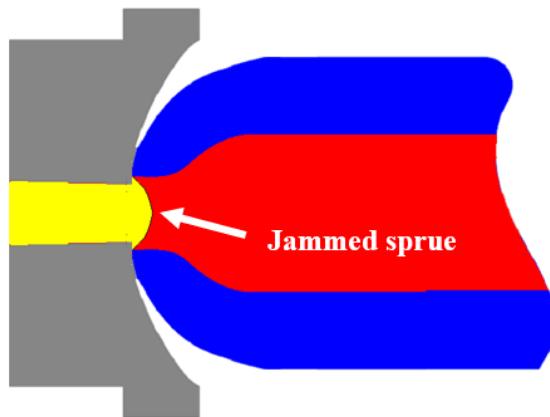
Before performing a **Universal MoldingTM** lab you should verify that the nozzle tip is properly sized. If it is incorrect, change it before starting.

Imagine what would happen if the diameter of the nozzle hole is larger than that of the bushing. A couple of defects could happen. One of them could be material degradation as a result of concentration of stress in the edges. That concentration of frictional stress would create degraded melt that could end up in the molded parts.



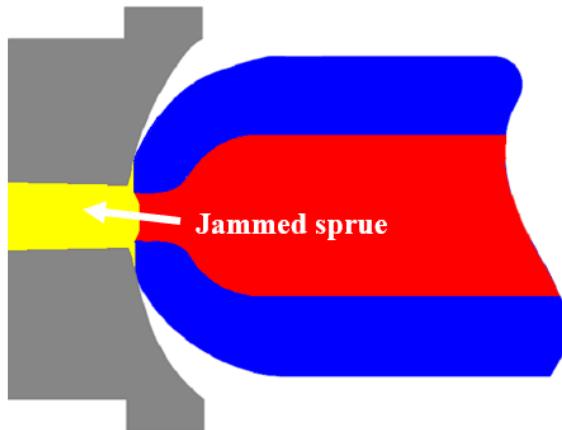
VI-28. Defect caused by stress concentration at the edge of the nozzle

Another defect seen in cold runner molds is that the sprue gets stuck in the nozzle. Remember that the sprue is part of the runner and should be ejected during demolding. In the example below, the sprue's solidified plastic has formed into the shape of a rivet head, which prevents it from ejecting.



VI-29. Defect caused by a sprue stuck in the nozzle

Now imagine what would happen if the radius of the nozzle tip is larger than that of the bushing. The melt could accumulate inside that gap and could cause the sprue to become stuck in place.



VI-30. Another defect caused by a sprue stuck in the nozzle

The sprue doesn't always get stuck as a result of inadequate coupling between the bushing and the nozzle. Although it's often the case, it's not the only reason. For example, it sometimes gets stuck because the interior surface of the bushing was polished in a direction perpendicular to that of the sprue's ejection. If that is the case, talk to the mold maintenance department and review the inner finish of the bushing.

This is why we prefer to say that the nozzle tip is part of the mold, even if it is installed in the injection unit. Some factories, when removing the mold for storage, also remove the nozzle and attach it to the mold.

Nozzle purge program

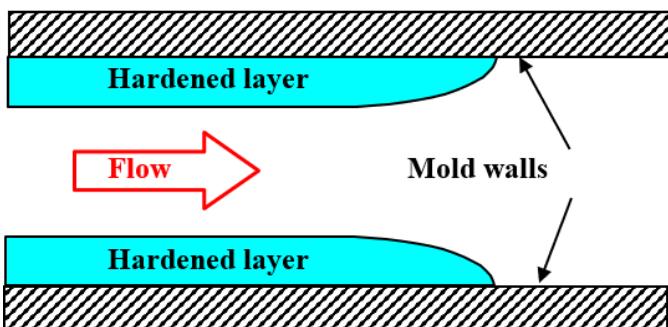
Some processes require special nozzle cleaning programs. This is seen more with molds for rigid PVC pipe joints. This material tends to be heat-sensitive and degrades easily. After each molding cycle, the injection unit pulls back, purging the small amount of plastic left in the nozzle.

This is done in order to remove any material remaining from the previous cycle. This purging would be used with single-cavity molds, usually large connectors. If the mold has multiple cavities, the degraded material will remain in the runner and will not need this program.

Never start a **Universal MoldingTM** lab without first installing the correct nozzle and sprue bushing combination.

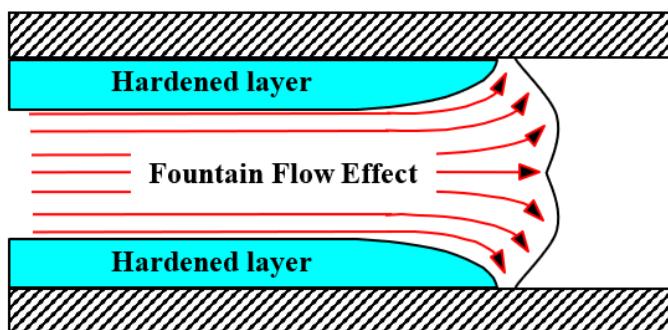
Fountain flow effect

The fountain flow effect describes the flow of a thermoplastic melt. Thermoplastic melt is compressible, opposes movement, and seeks to attach itself to the first static surface it finds. This opposition to flow is responsible for the fountain flow effect. Imagine melt flowing between the walls of a mold. The melt close to the metal surface of the mold will try to grab onto it and to any material already attached to that surface. That material in contact with the cold metal will create a layer of hardened material.



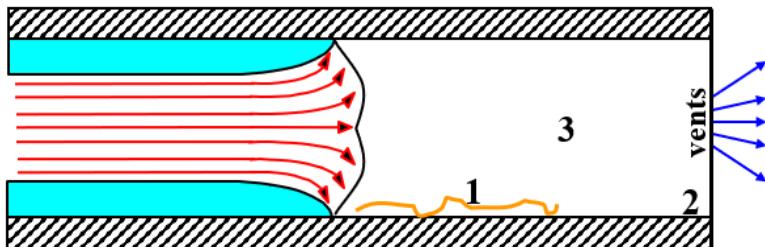
VI-31. Thermoplastic melt flow in a mold

The tendency would be to flow through the center in search of the opportunity to grab onto the first available static surface. It is this behavior that is responsible for the fountain flow effect that will be observed at the front of the melt flow. This means that the first melt that enters stays at the beginning of the mold.



VI-32. The fountain flow effect

Example: a piece of damp napkin is stuck to the surface of the cavity. The moisture is the only thing that is holding it. Once the mold closes, it is filled with melt in the indicated direction.



VI-33. Example of fountain flow effect using a piece of napkin

After filling, where will the paper napkin end up, position 1, 2, or 3?

It will stay perfectly flat on the surface where it was placed (position 1). Because of this effect, there is in-mold labeling technology.

Density and specific density

$$\text{Density} = \frac{\text{mass}}{\text{volume}}$$

$$\text{Specific density} = \frac{\text{material density}}{\text{water density}}$$

where the density of water is 1 g/cm³ at room temperature and pressure.

Dividing cancels the units, so if you ever find a specific density without units assume it is in g/cm³. If you want to work with other units, find the density of water with the preferred units and multiply it by the specific density.

Injection speed and injection flow

Today it is common to see more molders using **Universal** parameters. They prefer these as these represent what the mold sees. A cavity

represents a defined space or volume and is filled in an ideal time. From the mold's point of view, fill time is ***Universal*** and must be maintained. Imagine a scenario where you are forced to fill one less cavity in a mold, because the cavity was temporarily blocked for maintenance reasons. Apart from having to reduce the fill volume, you would also have to manipulate the injection speed to ensure the same injection time.

These days, it is common to see new machines provided with ***Universal*** parameters, such as injection flow instead of injection speed.

$$\text{Injection speed} = \frac{\text{distance}}{\text{time}}$$

$$\text{Injection flow} = \frac{\text{volume}}{\text{time}}$$

Injection flow (volume/time) is also ***Universal***. The relation between injection volume, area and screw diameter is:

$$\text{Injection volume} = (\text{screw area}) \times (\text{displacement})$$

$$\text{Screw area} = (\text{screw diameter})^2 * \frac{\pi}{4}$$

We substitute in the flow equation and get:

$$\text{Injection flow} =$$

$$\frac{(\text{screw diameter})^2 * \frac{\pi}{4} * (\text{displacement})}{\text{time}}$$

or

$$= (\text{screw diameter})^2 * \frac{\pi}{4} * (\text{injection speed})$$

These are simple equations used by machine controllers to change from machine parameters to ***Universal***. If your machine is not able to work with ***Universal*** parameters, use these equations and make the determinations yourself.

Barrel utilization

Barrel utilization ($\%U$) is a comparison between the maximum capacity of the injection unit and the capacity required to fill the mold.

$$\%U = \% \text{ of utilization}$$

$$\%U = \frac{(\text{volume used})}{(\text{volume the barrel is capable of})} * 100\%$$

Where:

volume used = the volume programmed to the mold's requirements
volume the barrel is capable of = the maximum recovery volume of the screw

Examples:

A barrel with a 35 mm screw, with a maximum recovery volume of 134 cm³, needs to recover 62 cm³. Since the machine's control is not **Universal**, what would be its recovery displacement?

Knowing that:

$$\begin{aligned}\text{Screw Area} &= (\text{screw diameter})^2 * \frac{\pi}{4} \\ &= (3.5 \text{ cm})^2 * \frac{\pi}{4} = 9.62 \text{ cm}^2\end{aligned}$$

Also, knowing that:

$$\begin{aligned}\text{Injection volume} &= (\text{screw area}) \times \text{displacement} \\ \text{Displacement} &= (\text{injection volume}) / (\text{screw area}) \\ &= (62 \text{ cm}^3) / (9.62 \text{ cm}^2) = 6.44 \text{ cm or } 64.4 \text{ mm}\end{aligned}$$

According to this calculation, the recovery displacement should be 64.4 mm (2.54 in).

How much would the percentage of utilization be?

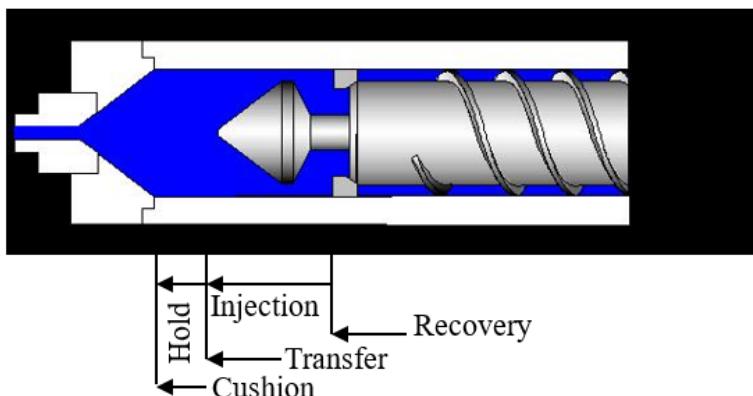
$$\%U = \frac{(\text{volume used})}{(\text{volume the barrel is capable of})} * 100\%$$

$$\%U = \frac{62cm^3}{134cm^3} * 100\% = 46\%$$

According to this calculation, the percentage of utilization is equal to 46%.

Are these recovery adjustment calculations reliable? They are a reference, since the final adjustment will be affected by the melt temperature, backpressure, leaks in the check ring, and the time it takes the check ring to seal.

Let's review:



VI-34. Fill positions

During recovery the screw rotates, forcing the melt to the front of the ring, pushing the screw back, and stopping when it reaches the desired recovery position. Once the signal giving permission to inject occurs, the screw will quickly inject until reaching the transfer position. Once the transfer position ends the hold stage begins; this is when the melt is compacted until all the cavities are completely filled. The final position of the screw is called the cushion.

In the recovery stage we fill the barrel with a homogeneous melt. This normally happens during the cooling stage. The control parameters are recovery time (or velocity) and backpressure. The recovery time should

never exceed the cooling time; this applies to injection units that are not provided with a shut-off valve in the nozzle.

Backpressure can be seen as a force that opposes free movement of the screw during recovery. With an increase in backpressure comes an increase in the amount of mass. Even if the recovery volume is the same, since the melt is compressible, the amount of material will increase. In addition, it increases mixing, friction, recovery time, and material degradation.

Backpressure has a significant effect on melt. Because of this, it should not be changed once it is optimized.

It is recommended keeping backpressure as low as possible. It should be increased only when more melting is required or when better mixing is needed. An initial/reference backpressure setting is 750 psi (5MPa) of plastic pressure. Remember that plastic pressure is the pressure that the melt sees.

Again, we emphasize that an increase in backpressure means an increase in recovery material as well.

Recovery position

The recovery position is the place the screw should reach in order to fill the mold. This is the result of combining the cushion, hold displacement, and injection displacement. In addition:

$$\begin{aligned} \text{Recovery position} = \\ \text{transfer position} + \text{injection displacement} \end{aligned}$$

In this equation the cushion is considered within the transfer position. The determination of the transfer position will be discussed later.

Combining the equations of weight, density, and volume we summarize:

$$\begin{aligned} \text{Injection displacement} &= \frac{1.27W}{D^2} \\ \text{Recovery position} &= \text{transfer position} + \frac{1.27W}{D^2} \end{aligned}$$

Where:

ρ = the melt's specific density (g/cm³)

W = weight of parts and runner (g)

D = diameter of the injection screw (cm)

This displacement is the distance between the recovery position and the transfer position. Note that this equation does not consider that, during injection, only about 95% of the mold is filled. This excess is ignored due to the fact that, during the injection stage, some material always sneaks to the other side of the check ring, whether during check ring closure or as a result of leaks between the check ring and the barrel.

It is imperative to clearly understand the effect of melt density during the determination of positions. During “molding from the desk”, initial screw positions are established, which may change during the process optimization labs. The density value provided by the raw material supplier can vary by more than 20%, since thermoplastic melts are compressible, and their density is influenced by melt pressure and temperature. During recovery, parameters such as back pressure and barrel zone temperatures affect this density. Additionally, determining the recovery positions becomes complicated if part of the melt passes to the other side of the check ring during injection. For this reason, optimization labs determine what ***Universal*** molders call discharge density, to correct these positions.

Discharge density

Discharge density is more precise for determining the recovery positions, since it considers several factors:

- mass
- volume
- melt temperature
- back pressure
- melt leakage through the check ring during injection

This density is calculated in an existing process by measuring the injected volume and the total injected weight. The injected volume is determined using the equation of a cylinder:

$$\text{Volume} = \text{area} \times \text{length}$$

Where:

$$\text{Area} = (\text{screw diameter})^2 * \frac{\pi}{4}$$

$$\text{Length} = \text{recovery position} - \text{cushion position}$$

To obtain this data in an existing process:

- The recovery and cushion positions can be obtained by navigating through the control pages.
- The injection weight is determined by weighing the molded parts plus the runner (if one exists).
- Finally, the discharge density is calculated by dividing mass by volume:

$$\begin{aligned} \text{Discharge density} \\ = \\ \frac{\text{total injection weight}}{((\text{screw diameter})^2 \times \pi/4) \times (\text{recovery position} - \text{cushion position})} \end{aligned}$$

If the discharge density is known, the injection displacement equation would look like this:

$$\text{Injection displacement}_{95\%} \times \text{E "desplazamiento de inyección"} = 95\% \frac{1.27W}{d^2}$$

$$\begin{aligned} \text{Recovery position} = \\ \text{transfer position} + 95\% * \frac{1.27W}{d^2} \end{aligned}$$

Where:

$$\rho_d = \text{discharge density (g/cm}^3)$$

$$W = \text{weight of the parts with the runner (g)}$$

$$D = \text{diameter of the injection screw (cm)}$$

Example:

There is a process with a demolding weight (parts and runner) of 17.5 grams, with recovery and cushion positions of 50.3 mm and 4.5 mm

respectively, and the injection unit has a 25 mm screw. What is the discharge density and the injection displacement₉₅?

Discharge density (g/cm³) =

$$\frac{\text{Shot weight}}{[D^2 \times \pi/4] \times [\text{recovery position} - \text{cushion position}]}$$

$$= \frac{17.5 \text{ g}}{\left[(2.5 \text{ cm})^2 \times \frac{\pi}{4}\right] \times [5.03 \text{ cm} - 0.45 \text{ cm}]} = \mathbf{0.78 \text{ g/cm}^3}$$

$$\text{Injection displacement}_{95} = 95\% \frac{1.27W}{\rho_d D^2}$$

$$= 0.95 \frac{1.27 \times 17.5 \text{ g}}{0.78 \frac{\text{g}}{\text{cm}^3} \times (2.5 \text{ cm})^2} = 4.33 \text{ cm} = \mathbf{43.3 \text{ mm}}$$

Note:

If you know the discharge density, ρ_d , then:

$$\text{Recovery position} = \text{transfer position} + 95\% \frac{1.27W}{\rho_d D^2}$$

This is because the discharge density considers the melt leaks during injection.

Recovery speed

Recovery speed is a parameter that we need to determine during molding from the desk. Currently, it can be adjusted in two ways:

1. revolutions per minute (rpm)
2. tangential speed (mm/s), which represents the linear speed of the screw along its circumference.

The relationship between these two parameters is defined as follows:

$$\text{tangential speed} = \text{rpm} \times \text{circumference}$$

$$\text{circumference} = 3.1416 \times D$$

$$D = \text{diameter of the injection screw (mm)}$$

The recovery speed should be adjusted so that recovery is completed before the expected cooling time ends. As a general rule, recovery should occur within 90% of the total cooling time. However, this speed must be determined empirically during the process, as the determination of rpm or tangential speed alone are not sufficient to ensure that the recovery will finish before cooling.

A more useful approach could be to calculate the recovery flow, which refers to the volume of material plasticized per unit of time. If we know the required volume to fill the cavity and the estimated cooling time, we can calculate the flow required to complete recovery with a 10% margin before cooling, using the following formula:

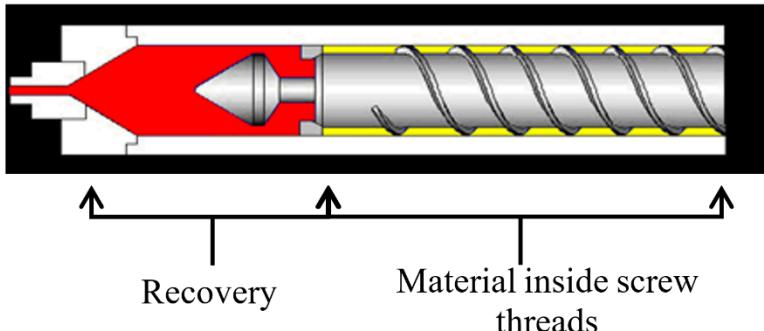
$$\text{Recovery flow} = \frac{\text{recovery volume}}{0.9 \times \text{cooling time}}$$

Unfortunately, many equipment manufacturers underestimate the importance of this parameter and do not incorporate it into their machine control systems.

Residence time

Residence time is defined as the time from which the material enters the injection unit to the time it is injected into the mold. In addition, if the mold includes hot runners, the time that the melt resides in the manifold should also be considered.

Another definition is that of the residence volume, the maximum amount of material that resides inside the barrel. Residence volume is greater than that of maximum recovery, since it also considers the volume of the material located between the screw threads.



VI-35. Residence volume

To determine the residence time, the volume of material residing in the screw threads, V_r , must be calculated. Determining the volume between the screw threads' complex geometry could be tricky. However, there is a simplified way to estimate V_r by assuming that it is 40% greater than the screw's maximum recovery capacity, V_{max} .

$$V_r = V_{max} \times 1.4$$

For example:

How much material lies in the screw threads of a 40mm barrel, which has a maximum injection capacity of 250.5 cc?

$$V_r = V_{max} \times 1.4 = 250.5 \text{ cc} \times 1.4 = \mathbf{350.7 \text{ cc}}$$

Residence time T_r is measured in cycles and is determined by rounding to the next whole number of:

$$T_r = 1.4V_{max}/V_{req}$$

Where:

V_{max} = maximum injection volume

V_{req} = volume required by the mold

The results of the equation for T_r is rounded to the next whole number, since decimals represent the fraction of uninjected melt that will reside in the barrel for an additional cycle, and not a fraction of a cycle.

The unit of residence time is cycles since we are molding from the desk and do not know the final duration of the cycle. Once molding is complete and we know the optimized cycle duration, it could be converted into seconds.

Another method with equal results is to use the % of utilization (%U).

$$T_r = 140 / \%U \text{ rounded to the next integer}$$

Where: $\%U = [V_{\text{required}} / V_{\text{max}}]100\%$

Example:

A 250.5cc barrel is used to mold two molds. One requires 150cc, the other 35cc. What would be residence T_r for each mold?

$T_r = 1.4V_{\text{max}}/V_{\text{required}}$	=	T_r (to the next whole num.)
$1.4(250.5) / 150$	2.3	3 cycles
$1.4(250.5) / 35$	10.02	11 cycles

VI-36. Table showing example of residence T_r

Or just use the following table:

$\%U$	T_r (# of cycles)
1%	140
2%	70
3%	47
4%	35
5%	28
6%	24
7%	20
8%	18
9%	16
10%	14
11%	13
12%	12
13%	11
14% - 15%	10
16% - 17%	9
18% - 19%	8
20% - 23%	7
24% - 27%	6
28% - 34%	5
35% - 46%	4
47% - 69%	3
>70%	2

VI-37. Table of residence time, according to % of utilization

%U based on industry type	Min	Max
General Molding	10%	80%
Precision	20%	65%
Optical Applications	20%	80%
High Speed	15%	40%
Reinforced with Fibers	20%	65%
Rigid PVC	20%	85%

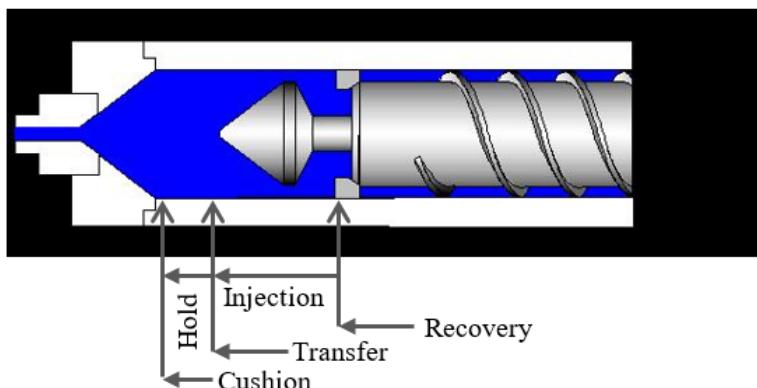
VI-38. Table showing % of utilization according to industry type

Examples:

- The high-speed industry typically has limited recovery times. Consequently, it will require a low $\%U$ (between 15% to 40%), or what would be equal to, for example, high melt residence (4 to 10 cycles), so that the plastic has enough time to melt.
- The heat-sensitive materials industry, such as those which use rigid PVC, demands a high $\%U$, or in other words, low residence times, to avoid material degradation.
- If the $\%U$ is very low you could have trouble controlling the cushion, as it would not have enough injection displacement to close the screw's check ring.

Transfer position

The transfer position is the position where the injection ends and hold begins. Remember that transfer is a screw position that corresponds to about 95% of the mold filled with compressible melt.



VI-39. Transfer position

Important facts:

- If the transfer position is too premature, the screw could create the bounceback effect.
- If the transfer position occurs too late, the screw can reach the end of the barrel, or rather zero cushion, and could create short pieces.

- What stops the screw after transfer is the melt in front of the check ring.
- From the barrel's point of view, the transfer position is reached when the injection volume is close to 95%. From the mold's point of view, at the instant at which the transfer position is reached, the volume that the melt occupies in the mold is less than what was injected. Remember that melt is compressible and will gradually expand.

After examining dozens of processes, interviewing multiple molders, and reviewing several manuals of various injection machines, the criteria used to determine transfer position was obtained through experimentation.

With machines under 400 metric tons

% of Utilization	35% or less	65% or higher	between 35% and 65%
Transfer	6 mm (0.25 in)	12 mm (0.5 in)	Interpolate

With machines larger than 400 metric tons

% of Utilization	35% or less	65% or higher	between 35% and 65%
Transfer	12 mm (0.5 in)	25 mm (1.0 in)	Interpolate

VI-40. Tables showing transfer position criteria

Once the transfer position has been obtained, convert it to volume (**Universal** unit).

$$\begin{aligned} \text{Transfer volume} &= \\ &\text{transfer position} \times \text{screw area} \\ &= \text{transfer position} \times (\text{screw diameter}^2 \times 3.142 / 4) \end{aligned}$$

The change must occur so that the screw's final position is greater than zero and less than the transfer position.

In the event of a defective mold requiring a transfer position other than the calculated one:

- if it is economically viable, repair the mold.
- if it is not, you will have to make the necessary changes and document them.

Examples:

1. A mold installed in a 160-ton press requires a fill volume of 120cc, and the barrel can be filled up to 160cc.

$$\text{Percentage of utilization} = (120/160) \times 100\% = 75\%$$

Since the % of utilization is greater than 65% and the press is less than 400 tons, the calculated transfer position would be 12mm.

2. A mold uses 50% of a barrel in a 200-ton press. To obtain the transfer position, you would have to interpolate between 6 and 12mm.
3. The recommended transfer position for a mold that uses 30% of the barrel would be 6 mm if the press is less than 400 tons and 12 mm if it is greater than 400 tons.

Summary

The screw positions are determined using the following:

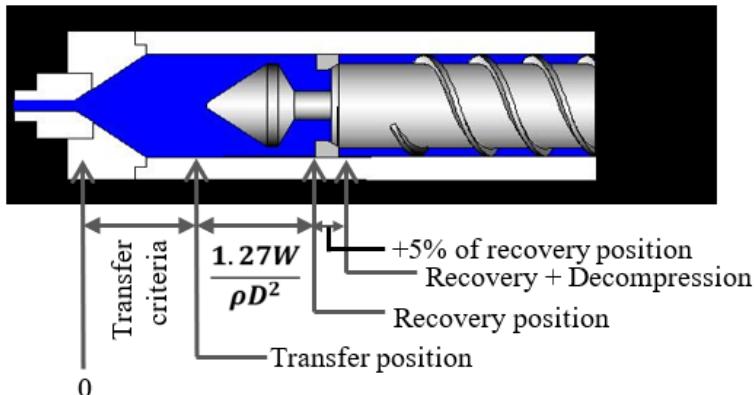
Transfer position = use the %U criteria

$$\text{Recovery position} = \text{transfer position} + \frac{1.27W}{\rho D^2}$$

or if discharge density is known, then use:

$$\text{Recovery position} = \text{transfer position} + 0.95 \frac{1.27W}{\rho_d D^2}$$

Recovery + Decompression = add 5% of recovery position



VI-41. Screw positions

Remember that you are molding from the desk, determining values that will be verified once the ***Universal MoldingTM*** lab is performed.

Temperature profiles

When we talk about temperature profiles, we mean the heat zones of the barrel. Heat zones are front, central, and rear.



VI-42. The barrel's heat zones

Each of these temperatures measures the steel temperature.

The main objective is to determine the temperatures before starting production to ensure optimal recovery and obtain the most significant temperature: that of the melt.

Resin suppliers typically recommend temperature ranges (upper and lower limits) for each barrel and melt zone.

To establish these temperatures, we consider three criteria:

1. Thin wall (***TW***):

The thin wall represents the filling difficulty. It is calculated as:

$$PF = (\text{farthest flow distance}) / (\text{thinnest wall in that flow path})$$

2. Percentage of Utilization (%U):

This compares the maximum barrel capacity with the capacity required to fill the mold:

$$\%U = (\text{volume used} / \text{barrel capacity}) \times 100\%$$

3. Cycle Duration:

This is the time between demolding parts. Long cycles are common in overmolding processes, where the cycle extends due to the removal of molded parts and the placement of a new insert, which prolongs the residence time of the melt in the barrel and the hot runner.

The most common criterion is as follows:

- If $\%U \leq 35\%$, use the lower limit for each zone.
- If $\%U \geq 65\%$, use the upper limit for each zone.
- If $35\% < \%U < 65\%$, interpolate between the limits.

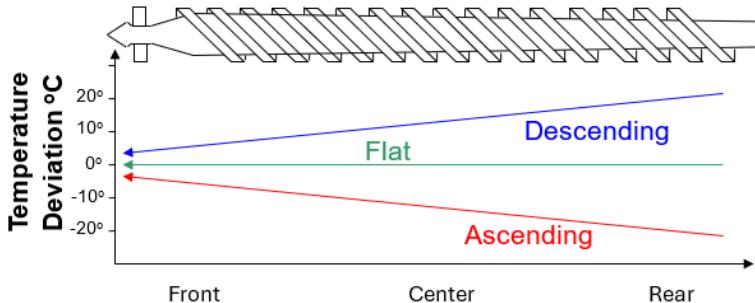
It is crucial to remember that the melt temperature is the most significant. If $PF \leq 100$, the melt temperature depends solely on $\%U$. If $PF \geq 200$, the melt temperature is increased to compensate for filling difficulty. If the cycle extends due to part removal and insert placement, it will be necessary to reduce the temperature to correct excessive melt residence.

Examples:

- Low $\%U$ ($\leq 35\%$): molding caps.
- High thin wall (PF) (≥ 200): molding tie wraps, which are thin and extremely long parts.
- Long cycles: In the case of molding parts with more than one resin, such as toothbrushes where the handle is made of polystyrene with sections containing an elastomer, cycles can extend during automatic switching between the different components.

In conclusion, we must be cautious when determining barrel zone temperatures, always prioritizing the melt temperature.

The temperature profile is typically recommended by the material manufacturer. There are three basic temperature profiles: descending, ascending, and flat.



VI-43. Temperature profiles

The vertical coordinate corresponds to the temperature deviation, while the horizontal coordinate corresponds to the heat zones.

Ascending profile - Temperatures go from low to high, and this profile is regularly used in situations where the residence time of the plastic is extremely high (% of utilization less than 30%), so that it avoids premature or extended heating of the material.

Descending profile – Temperatures go from high to low, and this profile is recommended for materials with additives and hard-to-melt materials. It is also recommended in operations where residence time is small, with a % of utilization greater than 65%.

Flat Profile – Here all temperatures are set the same. This could be used with materials with no fillings or additives, in which the residence time is moderate.

Remember that this is a reference, consult with your resin supplier for the most suitable profile for your material. Another source of information could be the screw and barrel supplier. For example, barrier screws are commonly used with highly semi-crystalline materials, and a descending profile would be recommended.

Because of the high variety of screw types and sizes, it is typical for most manufacturers to provide temperature ranges for each heat zone.

During the process of molding from the desk, we predefine the temperatures for each heat zone. We consider the factors mentioned

earlier, as well as material peculiarities. This includes aspects such as additives, friction sensitivity, melt difficulty, and sensitivity to high residence times.

Important:

- Thermocouples in the barrel's temperature zones read the temperatures of the metal.
- The most significant temperature is the temperature of the melt.
- Verify the temperatures with the resin supplier or the screw and barrel manufacturer.

Procedure for measuring melt temperature

1. Verify that the process has operated normally for at least ten cycles.
2. Preheat the measuring 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, retract the injection unit and purge the melt. You can do this on a removable surface for easier access.
4. Adjust the instrument to maintain the temperature's peak value. This eliminates subjectivity when searching for the melt stabilization temperature.
5. Submerge the instrument into the melt and stir it. When you notice the temperature starting to decrease, remove the instrument and note the peak temperature obtained.

Note:

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

Back pressure

Back pressure is applied to counteract the free movement of the screw during recovery, thereby generating additional pressure on the molten material. This increase in pressure manifests as friction, which, in turn, produces heat. As a result, the heater bands experience less demand since the friction significantly contributes to material heating. However, it is crucial to moderate back pressure, as excessive friction can lead to thermal degradation of the resin in certain materials.

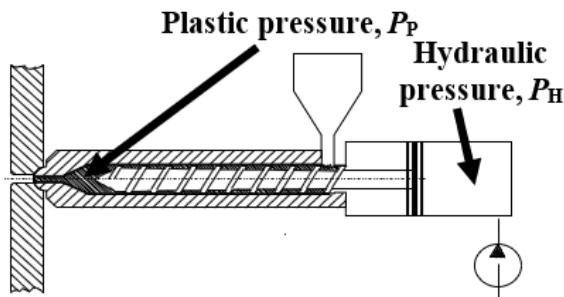
It's essential to exercise judgment because future changes in back pressure have multiple consequences. For example, when increased:

1. **Improved mixing:** Increasing back pressure can enhance the homogenization of additives in the melt.
2. **Degradation and breakage:** Sensitive materials may degrade, and fibers can break due to increased friction and heat.
3. **Equipment wear:** Higher back pressure can accelerate wear on the screw and barrel.
4. **Heat contribution:** With more back pressure, friction generates additional heat, potentially reducing the reliance of the heat from heater bands.
5. **Increased melt mass:** Molten thermoplastics are compressible; therefore, higher back pressure allows recovery of more material within the same volume. In other words, more material will be transferred to the mold during the injection stage.

Understanding these effects is crucial because adjustments to back pressure can have significant implications.

Intensification ratio (R_i)

It is imperative that the relationship between hydraulic pressure and plastic pressure be understood. Plastic pressure is intensified to a ratio of about 10 times the hydraulic pressure.



VI-44. Intensification ratio

The intensification ratio (R_i) represents the relationship between hydraulic pressure (P_H) and plastic pressure (P_p).

$$R_i = P_P / P_H$$

Other definitions are:

$$R_i = \frac{P_P}{P_H} = \frac{A_H}{A_P} = \frac{D_H^2}{D_P^2}$$

Where:

A_H = area of the hydraulic piston

A_P = area of the injection barrel

D_H = diameter of the hydraulic piston

D_P = diameter of the injection barrel

Important: Before molding, or making any adjustments to the injection machine, initial calculations must be done.

Machine labeling

Label your equipment with values that represent their capacities. This concept facilitates the understanding and use of your ***Universal*** factory.

Molding machine:

- maximum clamping force (US-ton, metric-ton, kN, ...)
- injection unit capacity (in^3 , cm^3 , liters, ...)
- maximum opening (in, mm, cm, ...)
- minimum opening (in, mm, cm, ...)
- space between tie-bars, horizontal (in, mm, cm, ...)
- space between tie-bars, vertical (in, mm, cm, ...)
- pattern of the ejectors on the platen (ex. 2x16in horizontal)

Dryer:

- air flow (CFM, liters/min, m^3/min , ...)
- drying hopper volume (ft^3 , liters, m^3 , ...)

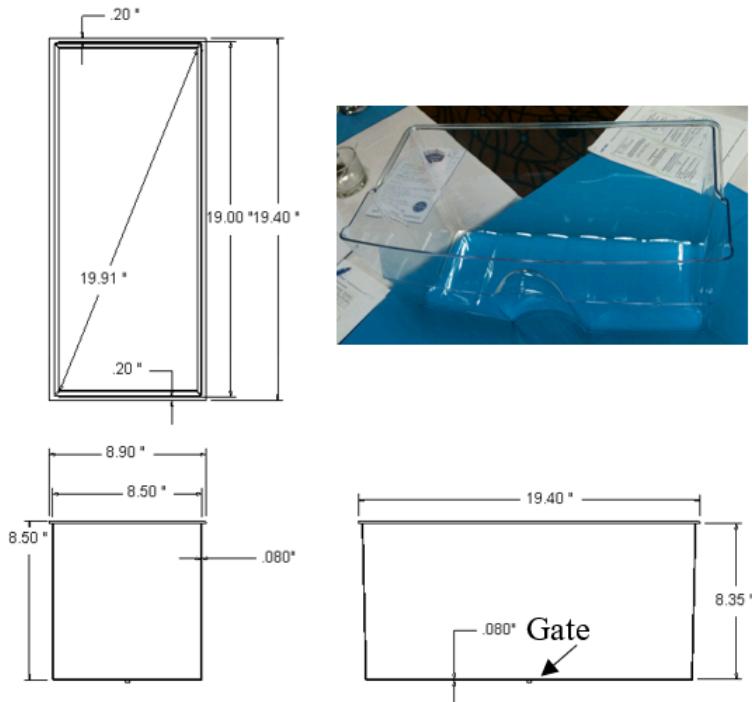
Water circulator:

- water flow (gpm @ psi, liters/min @ kPa, ...)

Questions

- 1) The nozzle hole is
 - a. greater than the diameter of the sprue bushing hole,
 - b. less than the diameter of the sprue bushing hole.
 - c. equal to the diameter of the sprue bushing hole.
- 2) The contact radius of the nozzle is
 - a. less than the contact radius of the sprue bushing.
 - b. greater than the contact radius of the sprue bushing.
 - c. same as the contact radius of the sprue bushing.
- 3) A material that degrades easily will be injected with a barrel larger than recommended. Since there is no smaller injection unit, what would be the recommended profile?
 - a. ascending
 - b. descending or flat
- 4) It is known that a fast process one that is less than 6 seconds. For a fast process with a material that does not degrade easily, it is recommended to use
 - a. a descending or flat profile with a larger barrel than normal.
 - b. an ascending profile with a barrel utilization of 70%.
 - c. a flat profile with 70% utilization.
- 5) What would be the recommended transfer position, from injection to hold, for a mold that uses 50% of the barrel in a 150-ton press?
 - a. Interpolating would give us a number between 6 and 12mm.
 - b. A number less than 6mm
 - c. A number greater than 12mm
 - d. 6mm
- 6) What would be the recommended transfer-to-hold position for a mold that uses 30% of the barrel?
 - a. Interpolating would give us a number between 6 and 12mm.
 - b. 12mm if the press is less than 400 metric tons
 - c. 6mm if the press is less than 400 metric tons, and 12mm if it is greater than 400 metric tons
 - d. 6mm

- 7) What would be the recommended transfer position for a mold that uses 50% of the barrel in a 150-ton press?
- Interpolating would give us a number between 6 and 12mm.
 - A number less than 6mm
 - A number greater than 12mm
 - 6mm
- 8) ABS, with density of 0.97 g/cc plasticized at 270°C and 50 bars, will fill a mold with a total weight of 147 gr. The ABS manufacturer recommends a barrel utilization between 40 and 60%. What would be the most appropriate barrel?
- $147\text{gr}/(0.97\text{gr/cc})$
 - $147\text{gr}/(0.97\text{gr/cc}) \times 52\%$
 - $147\text{gr}/(0.97\text{gr/cc}) \times 40\%$
 - between $147\text{gr}/(0.97\text{gr/cc})/60\%$ and $147\text{gr}/(0.97\text{gr/cc})/40\%$
- 9) Molding from the desk, a refrigerator drawer made of high impact polystyrene, in a single cavity mold with a hot sprue bushing, with:
- total shot weight = 1100 gr
 - 500-USton injection machine with a 90mm injection unit (2480cm^3).
 - material is high impact polystyrene (PS) with a density of 0.92 gr/cc.
 - PS cooling factor = 50lb/hr/ton of cooling
 - total expected cycle = 50 seconds
 - estimated cooling time = 15 seconds
 - drying data:
 - 100% virgin
 - drying time = 2 hours at 190°F
 - bulk density = 35lb/ft³
 - flow factor = 0.75 cfm/(lb/hr)
 - clamping force factor = from 1.5 to 2.5 USton/in²



- 9a. Determine the projected area in in^2 .
- 9b. Determine the thin wall value and force factor.
- 9c. Determine the required clamping force.
- 9d. Determine the total material consumption in lb/hr .
- 9e. Determine the volume required by the part in cm^3 .
- 9f. Determine the percentage of utilization of the barrel, $\%U$.
- 9g. Determine the cooling tons.
- 9h. Determine the water required from the chiller in gpm , assuming a Delta of 3°F .
- 9i. Determine the transfer position.
- 9j. Determine the recovery position.
- 9k. Recommend a plastic backpressure in psi .
- 9l. Recommend a decompression.
- 9m. What is the temperature, in $^\circ\text{F}$, of the recommended melt?
- 9n. Recommend a temperature profile for the heat zones of the barrel.
- 9o. Determine the residence time in cycles and seconds.
- 9p. Determine the volume of the drying hopper.

9q. Determine the flow required for the dryer in cfm.

- 10) A process demolds with a total weight (parts plus runner) of 18 grams, with a recovery position of 54.3mm, a cushion of 5mm, a transfer of 6mm, and an injection unit with a 25mm screw. What is the discharge density (ρ_d), the injection displacement₉₅, and the recovery position?

a. $\rho_d = \frac{18g}{[(2.5cm)^2 \times \frac{\pi}{4}] \times [54.3cm - 0.5cm]}$

$$\text{Injection displacement}_{95} = \frac{1.27 \times 18g}{\rho_d \times (2.5 cm)^2}$$

$$\text{Recovery position} = 6\text{mm} + 0.95 \times \text{injection displacement}_{95}$$

b. $\rho_d = \frac{18g}{[(2.5cm)^2 \times \frac{\pi}{4}] \times [54.3cm - 0.5cm]}$

$$\text{Injection displacement}_{95} = 0.95 \times \rho_d$$

$$\text{Recovery position} = 6\text{mm} + 0.95 \times \rho_d$$

c. $\rho_d = \frac{18g}{[(2.5cm)^2 \times \frac{\pi}{4}] \times [54.3cm - 0.5cm]}$

$$\text{Injection displacement}_{95} = 0.95 \frac{1.27 \times 18g}{\rho_d \times (2.5 cm)^2}$$

$$\text{Recovery position} = 6\text{mm} + \text{injection displacement}_{95}$$

VII. Machine Rheology

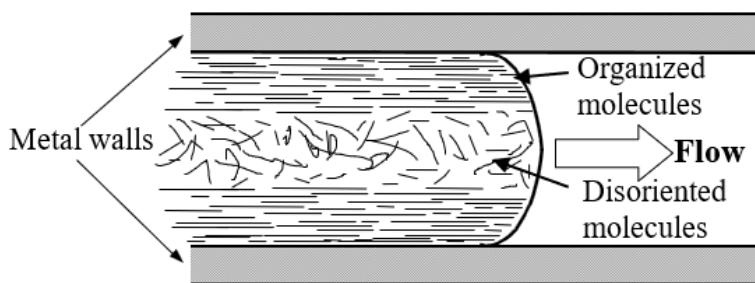
- Plastic Melt Flow
- Shear Stress, Viscosity and Shear Rate
- Machine Rheology by Power
- Machine Rheology by Viscosity
- Approximated Rheology
- Equation to Predict Injection Time

Plastic Melt Flow

The movement of material through a mold's cavities and its runners is better visualized if you imagine a flow of short, fine strings suspended in a melt. These strings are polymer chains that float freely, and their orientation depends upon the direction of the flow. These strings are molecules that conform to a specific orientation, which is determined by the flow once the material solidifies in the mold.

Organized molecules

The illustration shows the flow of melt through two fixed metal faces.



VII-1. Orientation of molecules in a melt flow

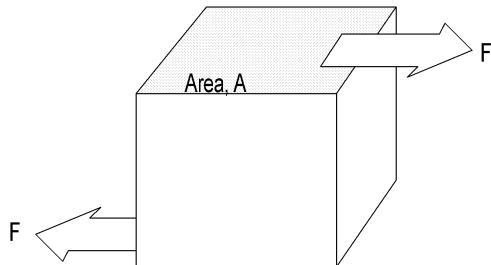
Molecules in contact with the metal tend to line up, and most stay in that direction after solidifying. The molecules that flow in the center are disoriented and remain that way after the melt solidifies.

The number of molecules that will remain oriented depends on how the polymer is cooled and its molecular weight. The longer the molecules or chains, the higher the molecular weight. The longer the molecules, the more difficult they are to orient and, consequently, the more they will restrict the melt flow.

Shear Stress, Viscosity and Shear Rate

The opposition to flow is better known as viscosity. Before defining viscosity, we must define other terms such as shear stress.

Shear stress can be defined as shear force per unit area. Imagine a cube with surface area, A , upon which two opposing forces of magnitude F try to tear the cube apart.



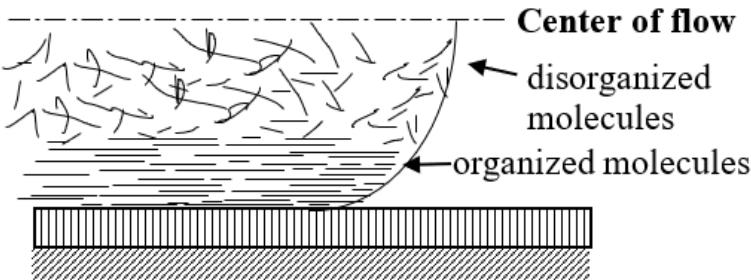
VII-2. Shear stress

This effect is defined as shear stress, τ .

$$\tau = \frac{F}{A}$$

Now try to imagine that the cube is a small mass of thermoplastic melt exposed to two opposing forces of magnitude F .

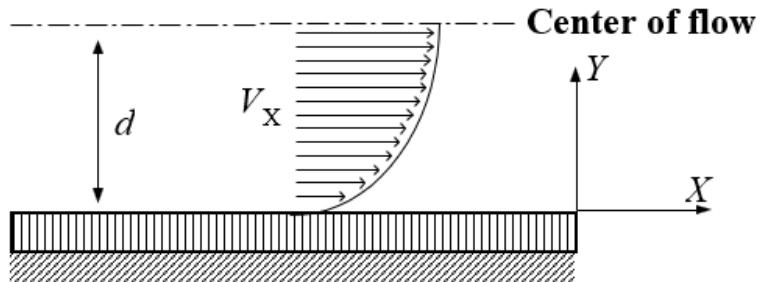
It was previously mentioned that thermoplastic melt resists flowing and will stick to the first static surface it finds along the way. It is this behavior that causes the shear effect of the melt. In order to simplify this explanation, let's consider only half the flow.



VII-3. Half of the melt flow

Likewise, the melt against the metal plate will experience an effort that opposes the movement of the melt. This effort is the result of disoriented

molecules opposing their orientation. Now let's set our coordinates; X is in the direction of the flow and Y is perpendicular to the flow.



VII-4. Velocity vectors in a melt flow

The molecules near the plate move at a different speed than those at the center of the flow. The speed profile V_x , in the direction of X , illustrates a maximum speed when $Y = d$ (center of the flow).

If you could microscopically see these molecules moving, you would see a gradual change of speed between both extremes. This velocity gradient in the vertical direction is defined as shear rate.

This change in speed V_x , in the direction of Y , is defined as:

$$\text{Shear rate} = \dot{\delta} = \frac{\text{change in speed}}{\text{distance}} = \frac{\Delta V_x}{\Delta Y}$$

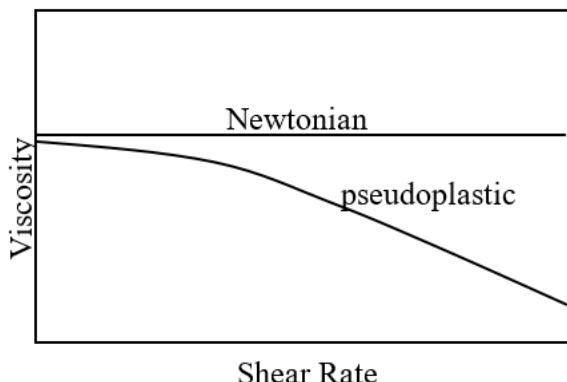
To describe the shear rate, imagine cars on a highway, one at normal speed (60 mph) being overtaken by other cars at an exaggerated speed (100 mph). Two scenarios, one car passes with a single lane separation and a second car with a separation of two lanes. In both cases, the change in speed (ΔV_x) is equal to 40 mph. However, when we consider the distance (ΔY) of the car passing with only one lane of separation, the wind effect will be more noticeable. The relationship between shear stress and constant change is viscosity, μ :

$$\begin{aligned}\text{Shear stress} &= \text{viscosity} \times \text{shear rate} \\ \tau &= \mu \times \dot{\delta}\end{aligned}$$

Viscosity (μ) can be displayed as the opposition to the flow. Understand that if the viscosity increases then the opposition to flow, or shear stress (τ), also increases.

Definition: a Newtonian flow is one in which viscosity is constant and independent of shear rate. The opposite would be a non-Newtonian flow, in which viscosity is a function of shear rate. Unfortunately, the molten plastic flow is non-Newtonian, which is why viscosity changes with shear rate. In a simplified form, *viscosity changes with injection speed*.

It has been experimentally demonstrated that viscosity decreases when the flow or injection speed increases. This behavior is called pseudoplastic melt flow, distinct from Newtonian flow in which viscosity is not affected by shear rate.



VII-5. Newtonian and pseudoplastic melt flow

This effect is called "shear thinning", increasing fluidity by friction. During the injection process the melt near the walls will harden, as a result of the heat exchange between the melt and the metal.

The flow in contact with the hardened layer will experience a resistance, resulting in an increase in friction. If this friction reflects in the form of heat, then we can say the following: if the speed increases, the friction increases; the heat will increase as well and, consequently, the viscosity will decrease.

Machine Rheology by Power

The plastic industry uses machine rheology to get the ideal injection time. We ***Universal*** molders use machine rheology by power, in which injection time and peak injection power are graphed.

Definitions:

Peak power – The maximum power reached by the injection unit, usually at the transfer position (change from injection to hold). This is obtained by multiplying the average injection flow by the pressure at the transfer position.

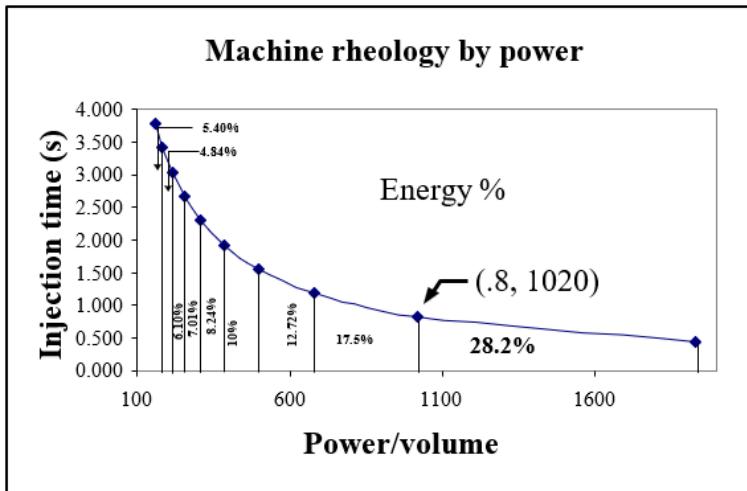
$$\text{Peak power} = \\ \text{average injection flow} \times \text{pressure at the transfer position}$$

Average injection flow – This flow is a function of the volume injected during the injection stage and the injection time.

$$\text{Average injection flow} = \\ \text{injection volume} / \text{injection time}$$

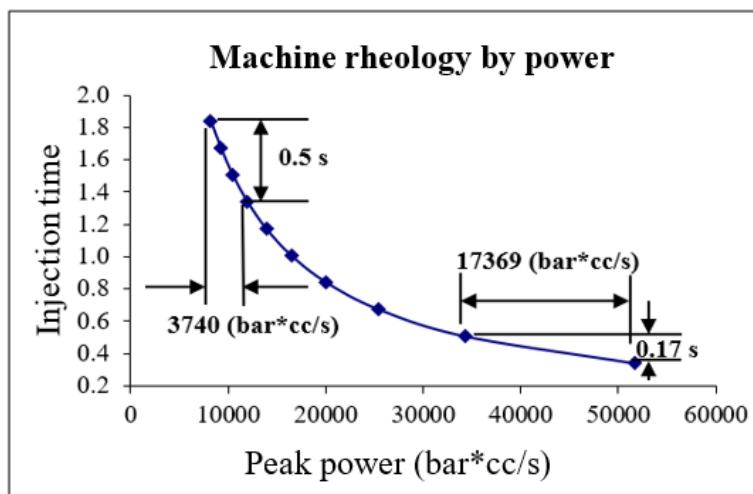
Injection volume – This represents the injected volume from the recovery position to the transfer position.

Injection time – This is the time it takes to inject from the recovery position to the transfer position. The injection time decreases with an increase in injection speed.



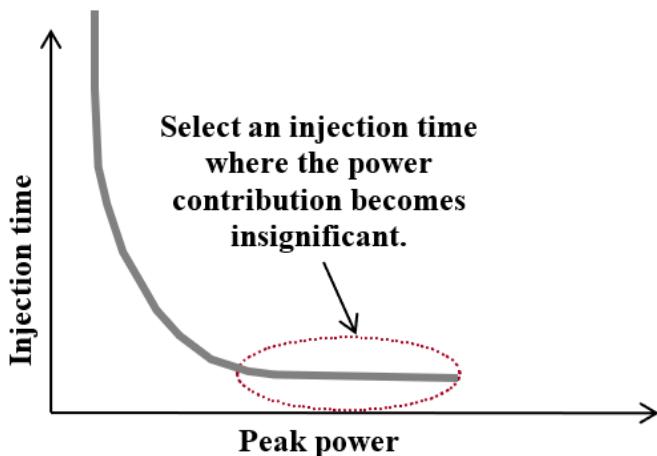
VII-6. Graph of machine rheology by power

The graph demonstrates the effect of the injection time on power, where the area under the curve represents the percentage of energy consumed by each decrease in injection time. This graph shows that the power increases when the injection time is decreased, or when the injection speed is increased. It also reveals that the energy consumed by the injection unit is more significant at low injection times.



VII-7. Graph of injection time versus peak power

In the above graph, with a slow injection, reducing the injection time by 0.5 seconds consumed 3740 bar*cc/s of power. Now, with a fast injection, reducing the injection time by 0.17 seconds consumed 17369 bar*cc/s of power, 4.6 times more power for a insignificant reduction in injection time.



VII-8. Zone where the change in injection time is minimal or where the power stopped contributing

Here, the objective is to select a point on the graph where the change in injection time is minimal or, rather, a point where the contribution of peak power becomes insignificant.

Machine Rheology by Viscosity

The plastic industry also uses rheology by viscosity to obtain the ideal injection time, using the equations previously defined with some assumptions.

$$\text{Shear stress} = \text{viscosity} \times \text{shear rate}$$

$$(\tau = \mu \times \dot{\delta})$$

$$\text{Intensification ratio, } R_i =$$

$$\text{plastic pressure, } P_p / \text{hydraulic pressure, } P_H$$

The intensification ratio was described previously and establishes the relationship between the pressure that the melt sees in the injection unit and the hydraulic pressure in the injection unit. These are related by a constant called the intensification ratio, R_i .

Plastic pressure (P_p) is the result of the stresses that oppose injection. From this premise it can be said that the plastic pressure P_p is apparently equal to the sum of all stresses opposing the melt flow entering the mold. This is why the plastics industry assumed:

$$\text{Plastic pressure } (P_p) = \text{relative shear stress}(\tau_R)$$

Substituting the intensification ratio equation, we get:

$$\begin{aligned} \text{Relative shear stress} &= \\ \text{hydraulic pressure} \times \text{intensification ratio} & \\ \tau_R &= P_H \times R_i \end{aligned}$$

Another assumed effect was simplifying the determination of shear rate ($\dot{\delta}$) by assuming that it was equal to the reciprocal of injection time (T), which was renamed relative shear rate ($\dot{\delta}_R$).

$$\dot{\delta}_R = \frac{1}{T}$$

Note that $\dot{\delta}$ and $\dot{\delta}_R$ are not equal, even though they have the same units (1/s). This simplification by the industry is based on unit cancellation; let me explain.

$$\text{Shear rate} = \dot{\delta} = \frac{\text{change in speed}}{\text{distance}} = \frac{\Delta V_x}{\Delta Y}$$

If we replace units (mm and seconds) in the shear rate equation and then cancel those units, we get:

$$\frac{\frac{\text{mm}}{\text{second}}}{\text{mm}} = \frac{1}{\text{second}}$$

Until now we are correct, the wrong thing to do would be to say that, according to this result (1/second), the equation is reduced to 1/time. The distance component of the numerator is in the direction of X and the distance component of the denominator is in the direction of Y , and those cannot be cancelled. For this reason, they use the word “relative”, and we advise against using the values of $\dot{\delta}_R$ in any formulation other than injection machine rheology. This is why we ***Universal*** molders prefer rheology by power.

If we substitute the definition of relative shear stress ($\tau_R = P_H \times R_i$) and relative shear rate ($\dot{\delta}_R = 1/T$) into the equation for viscosity ($\tau = \mu\delta$) and rearrange, we get:

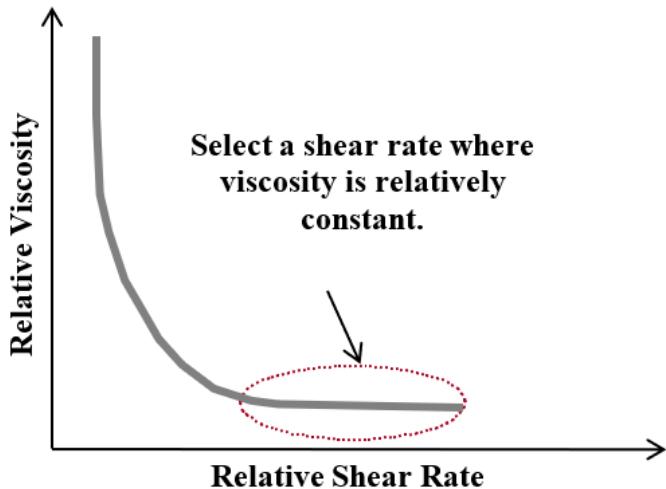
$$\mu_R = P_H R_i T$$

According to this equation, relative viscosity μ_R is simply determined by reading the hydraulic pressure and the injection time; then multiplying both by the intensification ratio.

Remember, if the injection unit reads plastic pressure and not oil (hydraulic) pressure, the equation would be:

$$\mu_R = P_P T$$

We then create a graph of viscosity versus shear rate and select a shear rate where the corresponding viscosity is relatively constant.



VII-9. Zone where the change in relative viscosity is minimal

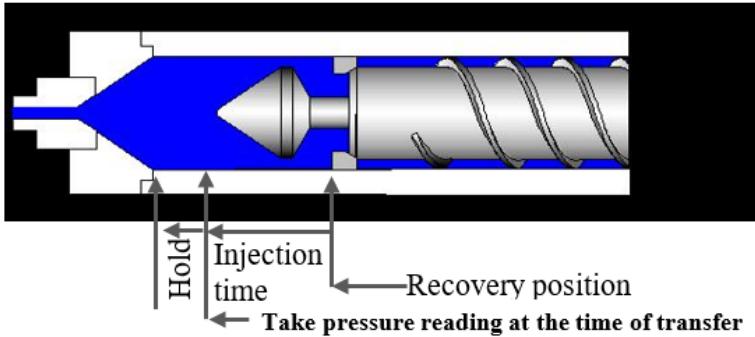
The idea is to select an injection time in which the melt is at its lowest viscosity and is relatively constant. The lower the viscosity the easier the melt flows and, as a result, the more efficient the fill.

Notes:

- These viscosity and shear rate equations should only be used in rheology for injection molding machines. Do not use these values for other calculations or other scientific work.
- If the injection unit provides the melt pressure, shear stress τ_R is equal to the pressure reading of the plastic melt, or $R_i = 1.0$.
- Although these equations do not represent the effects of viscosity and shear rate, they have standardized the determination of injection time. If you are not comfortable with these equations, use rheology by power.

Reading the values

Pressure is read at the time of transfer (change from injection to hold), and time is equal to the duration of the injection stage.



VII-10. Readings of transfer pressure and injection time

We use transfer pressure for convenience. The average pressure would be more representative; however, very few machines provide that reading.

These graphs are traditionally created by injecting at various injection speeds. For each injection speed, injection time and injection pressure readings are taken at the time of transfer. Then their corresponding coordinates are calculated.

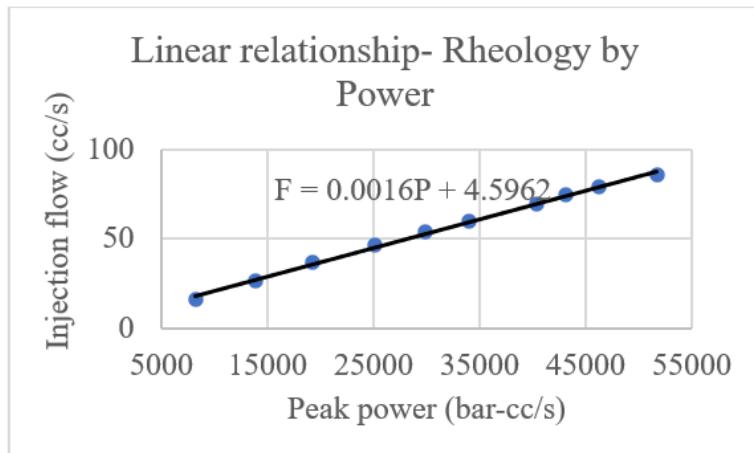
During the development of ***Universal MoldingTM***, it was established that the determination of the injection process parameters would use standardized procedures and corroborated equations. The next chapter will discuss the recommended procedures in detail.

Developing these graphs takes time and resources. We have developed a more efficient method called “approximated rheology”. Students from the University of Puerto Rico at the Mayagüez campus, under the supervision of Dr. Ivan Baigés, were the ones who initiated this simple technique.

Approximated Rheology

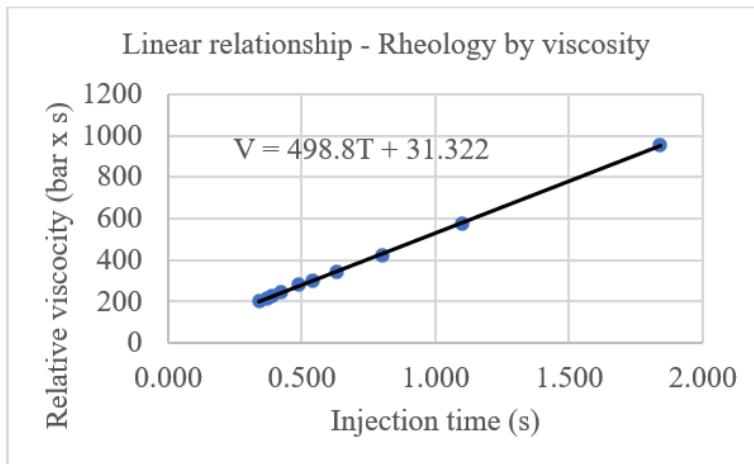
Developing a rheology lab in an injection machine consumes time and resources. With approximated rheology, a mathematical prediction technique, the laboratory is performed in less than a 3rd of the time. Let's see how.

If you work with rheology by power, you will notice that there is a relatively linear relationship between injection flow and peak power.



VII-11. Graph of linear behavior between injection flow and peak power

If you work with rheology by viscosity, there is a linear relationship between relative viscosity and injection time.



VII-12. Graph of linear behavior between relative viscosity and injection time

Both graphs reveal that the intermediate points on the graph can be approximated by the simple function of the equation of a line, $Y = Y_o + MX$. Let me explain, by knowing slope M and the intercept Y_o on the Y axis, we can predict intermediate points.

Example:

In the optimization of a process injecting a volume of 29.17cc, the injection time and transfer pressure readings were obtained at two injection speeds:

Vel (mm/s)	T_{inj} (s)	P_{tran} (bar)
193	0.34	602.7
19	1.84	518.5

VII-13. Example of injection times and transfer pressures

Using the equations shown above:

$$\text{Average injection flow} = \frac{\text{injection volume}}{\text{injection time}}$$

$$\text{Peak power} = \text{average injection flow} \times \text{pressure at the transfer position}$$

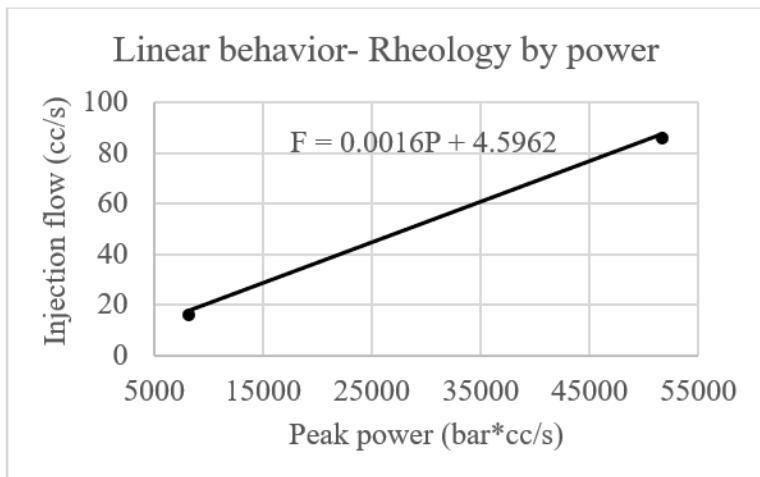
$$\text{Relative viscosity} = \text{plastic pressure} \times \text{injection time}$$

$$\text{Relative shear rate} = \frac{1}{\text{injection time}}$$

We calculate and add the values in the following table:

Inj. Vol. = 29.17 cc			Rheo. by Power		Rheo. by Viscosity	
Vel. (mm/s)	T_{inj} (s)	P_{tran} (bar)	Flow (cc/s)	Power (bar*cc/s)	V_{shear} (1/s)	Visc. (bar*s)
193	0.34	602.7	85.79	51708	2.94	204.9
19	1.84	518.5	15.85	8220	0.54	954.0

VII-14. Examples of rheology values



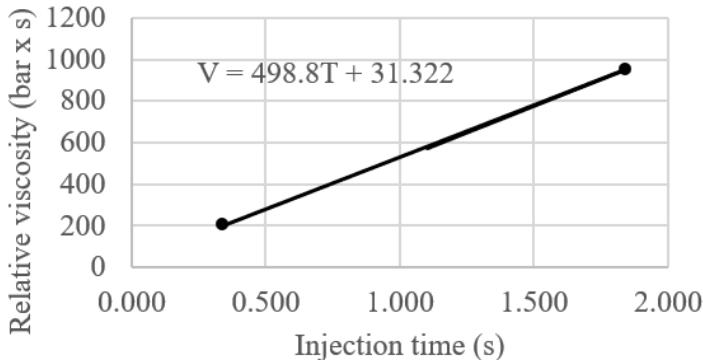
VII-15. Graph of the linear effect between average injection flow and peak power

The linear equation of rheology by power would be:

$$\begin{aligned} \text{Flow} &= (0.0016 \times \text{power}) + 4.5962 \\ \text{or} \\ \text{Power} &= (\text{flow} - 4.5962) / 0.0016 \end{aligned}$$

Now let's look at the linear equation of rheology by viscosity:

Linear behavior - Rheology by viscosity



VII-16. Graph of the linear effect between relative viscosity and injection time

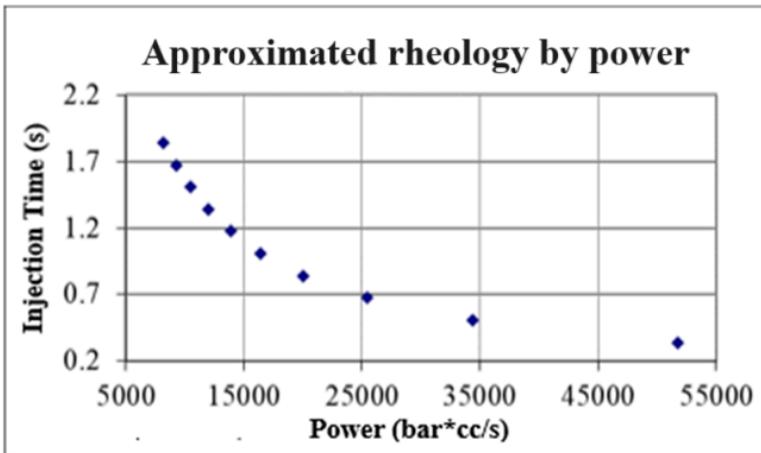
The equation would be:

$$\text{Viscosity} = (498.8 \times \text{injection time}) + 31.322$$

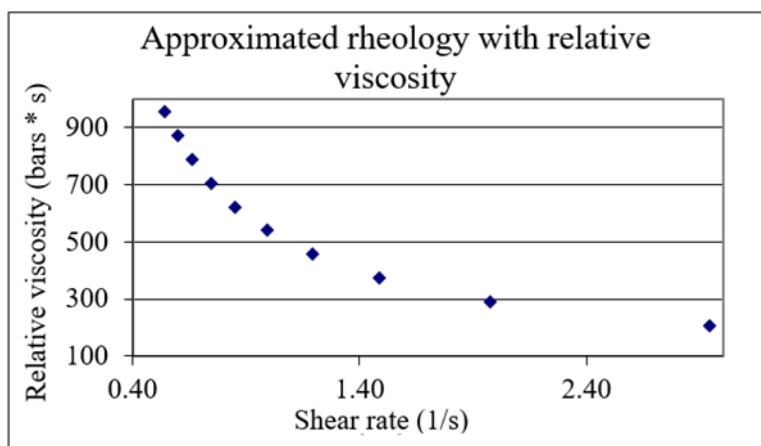
The next step is to determine eight injection times equidistant between the existing times, T_{inj} , then complete the calculation of the corresponding average injection flow and shear rate for each injection time added.

Inj. Vol. 29.17 cc			Rheo. by power		Rheo. by viscosity	
Vel (mm/s)	T_{inj} (s)	P_{Tran} (bar)	Flow (cc/s)	Power (bar*cc/s)	V_{shear} (1/s)	Visc. (bar*s)
193	0.34	602.7	85.79	51708	2.94	204.9
	0.51		57.57	34337	1.97	284.0
	0.67		43.32	25430	1.49	367.2
	0.84		34.73	20058	1.19	450.3
	1.01		28.98	16465	0.99	533.4
	1.17		24.86	13892	0.85	616.6
	1.34		21.77	11960	0.75	699.7
	1.51		19.36	10455	0.66	782.8
	1.67		17.43	9249	0.60	866.0
19	1.84	518.5	15.85	8220	0.54	954.0

VII-17. Table including 8 equidistant injection times

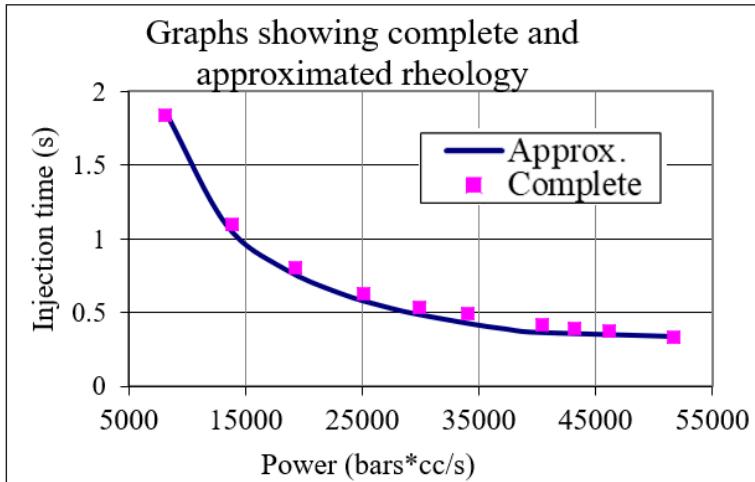


VII-18. Graph of approximated rheology by power



VII-19. Graph of approximated rheology using relative viscosity and shear rate

Now let's superimpose the graph of an approximated (two-point) rheology with the graph showing a complete (ten-point) rheology.



VII-20. Superimposed graphs of complete and approximated rheology

This graph shows that approximated rheology is very close to complete (ten-point) rheology, which makes it a timesaving, beneficial option.

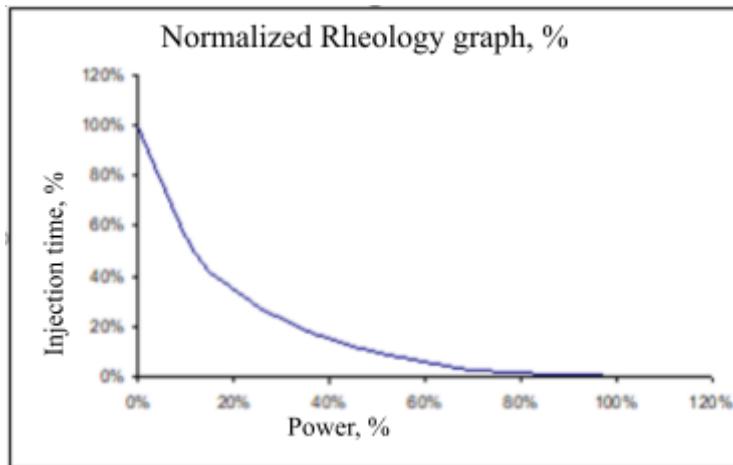
Again, we ***Universal*** molders prefer rheology by power because the definitions describe the effect with more conviction. You can use the method which makes you feel more comfortable; both rheologies lead to good results.

Equation to Predict Injection Time

During the development of ***MUTM*** it was established that all parameters must be determined by clear procedures and equations. This was done to guarantee similar results between molders.

If the selection of injection time depended on individual criteria, then it would be impossible to obtain similar results between molders. Using the criteria of multiple molders, who have a clear understanding of machine rheology, this situation was resolved. With rheology graphs with their coordinates normalized in percentages and simple statistics, the selection trend of some ***Universal*** molders was determined.

With several graphs such as the one illustrated, the following question was formed: Where does the plateau begin?



VII-21. Graph of standardized rheology in %

The average of this entire group was evaluated, and the following formula was obtained:

$$T_{plateau} = T_{min} + (T_{max} - T_{min}) / 9$$

Where:

- T_{min} - injection time corresponding to the maximum injection speed
- T_{max} - injection time corresponding to the minimum injection speed
- $T_{plateau}$ - injection time where the plateau starts on the graph

This equation calculates injection time, where the rheology curve begins to be relatively constant. Based on this simple calculation, we determine that the ideal injection time should be less than that obtained by the equation.

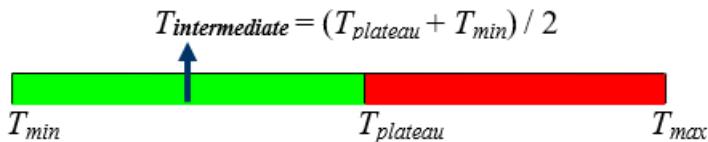
$$T_{ideal} < T_{min} + (T_{max} - T_{min}) / 9$$

The ideal injection time will depend on the type of industry. Conventional molding industries should select an ideal injection time in the center of the plateau. Industries that mold materials sensitive to friction as a result

of high injection rates, such as the rigid PVC fittings industry, should be close to $T_{plateau}$. High-volume injection industries, such as cap manufacturers, should be close to T_{min} .

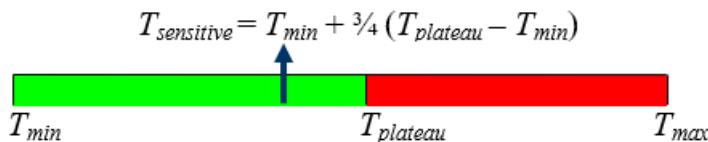
Let's take a closer look:

Conventional molding industries: Select an ideal injection time between $T_{plateau}$ and T_{min} .



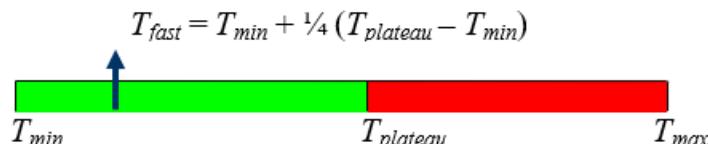
VII-22. Injection time for conventional molding industries

Industries that mold friction-sensitive materials (such as manufacturers of rigid PVC fittings): For materials that degrade at high speeds, select an ideal injection time near $T_{plateau}$.



VII-23. Injection time for industries that mold friction-sensitive materials

Industries with a high volume of production (such as cap manufacturers): Select an ideal injection time near T_{min} .



VII-24. Injection time for high-volume injection industries

T_{fast} is also used with parts that are very difficult to fill, like in micromolding.

If we replace the definition of the plateau time ($T_{plateau} = T_{min} + (T_{max} - T_{min}) / 9$) within each of the three previous equations and simplify, we get:

Conventional molding industries

$$T_{intermediate} = T_{min} + (T_{max} - T_{min}) / 18$$

Industries that mold sensitive materials

$$T_{sensitive} = T_{min} + (T_{max} - T_{min}) / 12$$

High-volume injection industries

$$T_{fast} = T_{min} + (T_{max} - T_{min}) / 36$$

After selecting the application type, select the corresponding equation, replace the values of T_{min} and T_{max} , and determine the ideal injection time.

Example: Using the calculated rheology for $T_{min} = 0.24$ seconds and $T_{max} = 1.58$ seconds, the corresponding injection times would be:

Average application:

$$\begin{aligned} T_{intermediate} &= T_{min} + (T_{max} - T_{min}) / 18 = 0.24 + (1.58 - 0.24) / 18 \\ &= \mathbf{0.31 \text{ seconds}} \end{aligned}$$

Sensitive material application:

$$\begin{aligned} T_{sensitive} &= T_{min} + (T_{max} - T_{min}) / 12 = 0.24 + (1.58 - 0.24) / 12 \\ &= \mathbf{0.35 \text{ seconds}} \end{aligned}$$

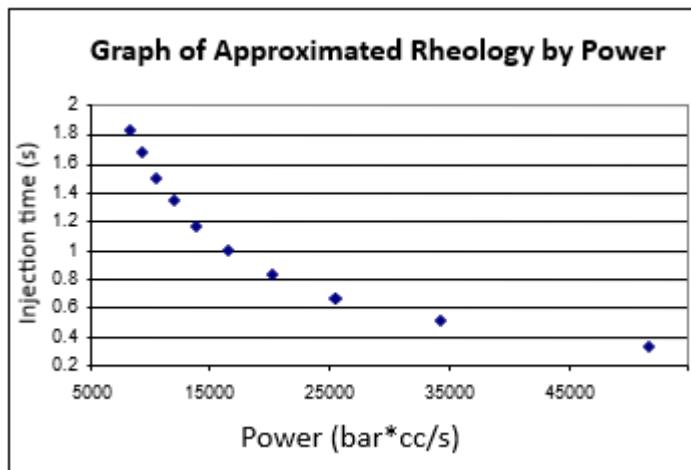
High volume application:

$$\begin{aligned} T_{fast} &= T_{min} + (T_{max} - T_{min}) / 36 = 0.24 + (1.58 - 0.24) / 36 \\ &= \mathbf{0.28 \text{ seconds}} \end{aligned}$$

Procedures for obtaining these minimum and maximum injection times will be discussed later.

Questions

- 1) Using calculated rheology, you can determine the ideal injection time without having to create a graph. For example, depending on the type of industry, the injection time should be less than $T_{min} + (T_{max} - T_{min})/9$. For a T_{min} of 0.24 seconds and a T_{max} of 1.58 seconds, the ideal injection time should be
- Shear rate = $0.24 + (1.58 - 0.24)/9$
 - $T_{ideal} < 0.24 + (1.58 - 0.24)/9$
 - 4 seconds.
- 2) For a process with a total cycle of 3.5 seconds, we use the approximated rheology equation
- $T_{intermediate} = T_{min} + (T_{max} - T_{min})/18$
 - $T_{sensitive} = T_{min} + (T_{max} - T_{min})/12$
 - $T_{fast} = T_{min} + (T_{max} - T_{min})/36$
- 3) In the graph of approximated rheology by power below, an ideal injection time would be
- 1 second.
 - greater than 1 second.
 - 0.4 seconds.

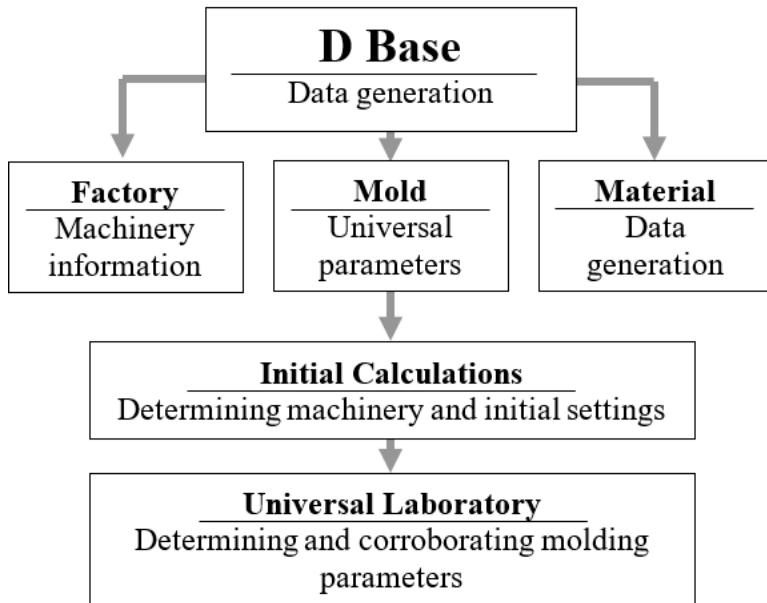


- 4) ***Universal*** molders prefer machine rheology by power
- a. because shear rate is graphed versus injection power.
 - b. because it is graphed with parameters that truly describe the effect being measured, injection time versus peak power.
 - c. because viscosity versus injection power is graphed here.
- 5) Peak power
- a. is calculated by multiplying average injection time by the pressure at the time of transfer.
 - b. is the maximum plastic pressure reached by the injection unit, usually at the time of transfer.
 - c. is obtained by multiplying the average injection flow by the pressure at the time of transfer.
- 6) Average injection flow
- a. is a function of injection time and the volume of melt filled in the injection stage.
 - b. is the maximum plastic pressure reached by the injection unit, usually at the time of transfer.
 - c. is obtained by multiplying the average injection flow by the pressure at the time of transfer.
- 7) The injection volume represents the volume filled from the recovery position to the transfer position.
- a. True
 - b. False

VIII. Determining Injection Speed

- **Laboratory I - Understanding Fill and Its Limitations, Determining Minimum Injection Time and Injection Pressure Limit**
- **Laboratory II - Procedure for Determining Minimum Injection Time and Injection Pressure Limit**
- **Laboratory III – Determining Injection Time and Injection Speed**
- **Laboratory IV - Graph of Rheology and Determination of Ideal Injection Time and Speed**
- **Laboratory V - Approximated Graph**
- **Laboratory VI - Injection Time Prediction**

In order to maximize the use of this molding discipline, you must have a clear understanding of the fundamentals of ***Universal MoldingTM*** and "Molding from the Desk". Understand and identify the needs of your process, and address those needs with well thought-out solutions. ***Universal MoldingTM*** is a discipline that promotes a structure of organized events.



VIII-1. Flow chart of ***Universal MoldingTM*** events

Collect all the information, organize a database, and make that information accessible to everyone in your molding factory.

Include the following:

Molding machine data:

- minimum and maximum clamping forces
- ejector pattern
- space between the bars
- minimum and maximum mold width
- maximum mold weight
- injection volume
- maximum injection pressure

- intensification ratio
- special options such as sprues, nozzle valves, etc.

Auxiliary equipment data:

- dryer (drying flow, hopper volume, ...)
- TCU (flow, pressure, ...)
- additive feeder
- grinders
- robots
- conveyors
- hot runner control, etc.

Mold data:

- ejector pattern
- dimensions and opening
- weight
- fill volume
- hot runner
- operational limits
- material or materials to be molded
- mold temperatures
- water flow and pressure drop
- diameter and radius of the sprue bushing, etc.

Data of the material being used:

- name of the material and its distributors
- specific melt density
- specific density to environmental conditions
- bulk density of resin
- if hygroscopic, the drying time and temperature
- melt temperature
- suggested barrel temperature profile
- suggested mold temperature
- semi-crystalline or amorphous
- suggested plastic injection pressure, etc.

Once you have all the data of your process, complete your "Molding from the Desk", performing the calculations for drying, the press, the injection unit, cooling, etc.

Finally, determine the optimal molding parameters by performing the ***Universal MoldingTM*** procedures.

Again, before proceeding with this ***Universal MoldingTM*** laboratory remember that:

- You must perform the "Molding from the Desk".
- All auxiliary equipment should be properly installed and operational.
- Temperatures should already have been reached, such as water temperature, injection barrel temperature profile and corresponding melt temperature, hot runner heat zones (if any), etc.
- Barrel adjustments should have been programmed, such as recovery and transfer position, decompression, recovery speed, etc.
- The appropriate nozzle tip should be installed.
- The required clamping force should be set.
- The opening of the platens, their movements, speeds and the mold protection must be carefully and precisely set.
- Extended cooling time must be programmed. Remember that this has to be set to more than is required. To prevent it from interfering with previously determined parameters, it will be optimized at the end.

Important -- Only qualified personnel who have read the operational manuals of the equipment and understand the functionality of the equipment should operate and/or adjust them.

Laboratory I - Understanding Fill and Its Limitations, Determining Minimum Injection Time and Injection Pressure Limit

The goal of this lab is to understand the fill's behavior, determining how fast the mold can be filled without causing defects. The maximum speed is determined, its corresponding maximum injection pressure is obtained, and fill limitations, if any, are identified. These fill limitations could be:

- Material degradation or burning as a result of excess speed. For example, PVC tends to burn if the injection speed is high.

- A problem with the vents, such as burns that can occur due to gas combustion, or dieseling. This defect can be corrected by cleaning the vents or, in the worst case, repairing or modifying the mold.
- Equipment limitations, such as an injection unit unable to reach high injection speeds, a limitation that could be a result of an injection unit that is inadequate for the mold, etc.

The idea is to identify in advance any defects or limitations that may arise. If any of these occur in the equipment when increasing the injection speed, you must decide whether the modification is simple or complicated. If the remedy is simple, do it. However, if the modification or change is not viable, or is not economically feasible, you will have to carry out the laboratory with that condition. Ideally it would be better to correct the situation before continuing; unfortunately, it is not always possible. For example, in the case of limited injection speed, it would be better to switch to an appropriate injection unit, but if that is economically prohibitive, you will have to work with a limited injection speed.

It is important to understand that not every parameter programmed in the equipment's control can be reached; therefore, you must ensure that the entered value is being achieved.

The programmed speed may be limited by an inadequate injection unit, as a result of a low % of utilization. For example, a utilization of 5% might not have the acceleration displacement required by the speed that was entered. Verify that the programmed speed is always being reached.

Procedure for Determining Minimum Injection Time and Injection Pressure Limit

- a) Verify that the transfer position from injection to hold was entered. Do you remember how? This procedure was explained in the "Injection Unit Calculations" section.

With machines under 400 metric tons

% of Utilization	35% or less	65% or more	Between 35% and 65%
Transfer	6 mm (0.25 in)	12 mm (0.5 in)	Interpolate

With machines 400 metric tons or larger

% of Utilization	35% or less	65% or higher	Between 35% and 65%
Transfer	12 mm (0.5 in)	25 mm (1.0 in)	Interpolate

VIII-2. Tables to select the injection-to-hold transfer position

- b) Turn off the hold stage so that it does not interfere with the determination of the minimum injection time. This could be done by setting the hold pressure to zero. Some injection machines come with the hold stage divided into two, pack and hold. Turn off one of them.
- c) Adjust the injection unit to produce parts that are 20% incomplete. This is done to prevent damage to the mold and the machine.

$$\begin{aligned} \text{Recovery position} = \\ \text{transfer position} + 80\% \text{ of injection displacement} \end{aligned}$$

Notes:

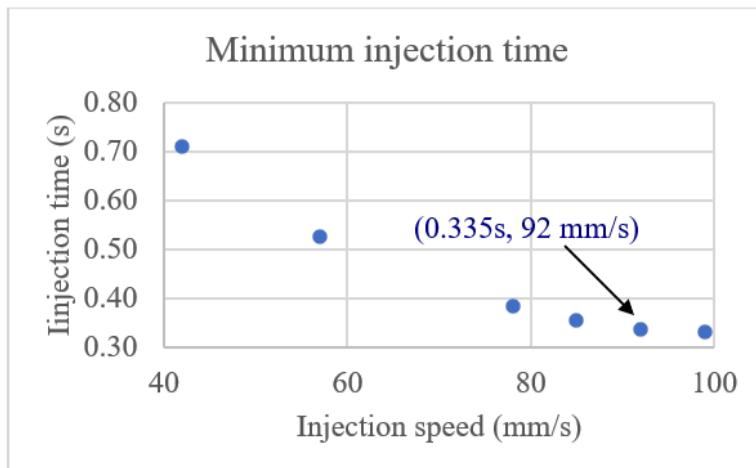
- It is important to maintain a cooling time that is longer than required while determining the minimum injection time.
 - You should also be alert during mold closure, to prevent the mold from closing with unreleased parts.
 - Turn off alarms that could stop the laboratory, such as the alarm that limits the maximum injection time. Caution: never turn off mold closure protection (*mold protect*).
- d) Find the mold's minimum injection time and maximum injection speed. The idea is to increase the injection speed until the injection time stops decreasing. A safe and practical method is to increase the injection speed and injection pressure limit at the same time, until the

injection time stops decreasing. Be cautious and, even if it's easier, don't set the pressure limit to its maximum. It may cause breakage.

During the experiment:

- Verify that parts do not remain trapped in the cavities. You may need to perform this step with the control set to semi-automatic.
- If you turned off the hold stage by adjusting the hold time to zero, verify the demolding. If the parts are extremely hot, increase cooling time until the parts demold at a lower temperature.

It is advisable to perform this experiment with a graph of injection time versus injection speed, which will help you determine the minimum injection time. See below.



Pressure (mPa)	Time Inj. (s)	Vel (mm/s)
-----	-----	7
-----	-----	14
-----	-----	21
-----	-----	28
-----	-----	35
151	0.711	42
-----	-----	50
166	0.526	57
-----	-----	64
-----	-----	71
189	0.384	78
196	0.354	85
204	0.335	92
211	0.330	99

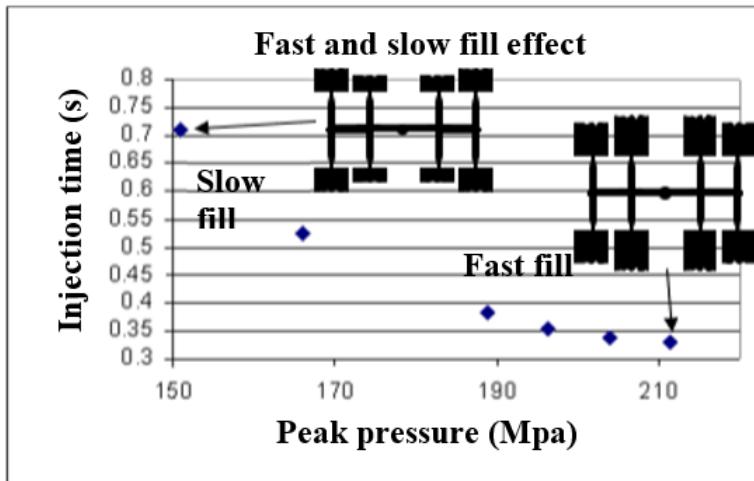
VIII-3. Determination of minimum injection time and maximum injection pressure (example)

The graph of injection time versus injection speed illustrates how, at a pressure of 204 mPa and at 92mm/second, the decrease in time becomes insignificant.

- e) Inspect the parts. Although they will be incomplete, make sure you obtain parts that are free of defects due to degradation.

If the defect requires a simple mold repair, do it and repeat the exercise. However, if the repair is not feasible, the speed at which the defect appeared should be your maximum speed.

Observation: the fill of the parts increases with an increase in injection speed.



VIII-4. Effect of injection time on parts fill

This is a normal effect that will not affect the experiment; a slow fill produces incomplete parts and a fast fill will produce parts that might appear to be totally full. Even though the recovery position was adjusted to 80% of the fill (parts and runner), the parts could fill more than 80% with fast fillings.

Remember that at the end of the injection, transfer position, the screw is being stopped by the plastic in front of the check ring. The screw, just like the plastic, will continue to travel with its own kinetic energy. In other words, the melt will continue to fill the mold until it consumes all that energy in the form of speed. That is why the higher the injection speed the higher the energy at transfer and, consequently, the higher the fill volume.

Continue determining the minimum injection time.

f) Write down the maximum pressure and the maximum speed obtained.

$$V_{max} = \text{maximum speed}$$

$$PH_{max} = \text{maximum hydraulic pressure}$$

If you are working with ***Universal*** parameters, it will be:

$$\begin{aligned}F_{max} &= \text{maximum flow} \\PP_{max} &= \text{maximum plastic pressure}\end{aligned}$$

Summary of the steps:

- a) Adjust the transfer position to hold.
- b) Turn off the hold stage so that it does not interfere with the determination of the injection time.
- c) Adjust the injection unit to produce parts that are 20% incomplete.

$$\begin{aligned}\text{Recovery position} &= \\&\text{transfer position} + 80\% \text{ of injection displacement}\end{aligned}$$

- d) Find the minimum injection time and maximum injection speed for the mold being used.
- e) Although the parts will be incomplete, verify they are free of defects due to degradation.
- f) Write down the maximum pressure and the maximum speed obtained.

$$\begin{aligned}V_{max} &= \text{maximum speed or } F_{max} = \text{maximum flow} \\PH_{max} &= \text{maximum hydraulic pressure or} \\PP_{max} &= \text{maximum plastic pressure}\end{aligned}$$

Notes:

- Disabling the hold stage by setting pressure or time to zero can be challenging on some machines because they do not allow precisely zero values. Additionally, certain machines may become unstable if the hold pressure is set to zero. It is essential to have a clear understanding of your machine's controls or consult with its manufacturer if you have any doubts.
- If any equipment limitation or defect arises, evaluate the situation, and if the limitation requires a modification or a simple change of equipment, do so. However, if the modification or change is not feasible, you will have to carry out the laboratory with what you have.
- The programmed injection speed might not be reached by the injection unit; verify that it does.
- This lab is performed with the hold stage off. You could do this by adjusting the hold pressure or time to zero.

- In order to avoid damage to the mold and the machine, this laboratory is carried out by adjusting the injection unit to produce parts that are 20% incomplete.
- The weight of the parts increases with an increase in injection speed or flow.

Laboratory II - Determination of Injection Time and Injection Speed

- a) Set the injection speed or speeds to 95% of V_{max} , the maximum speed found and call it V_{95} .

$$V_{95} = 0.95 \times V_{max}$$

Use only one fill speed. Previous chapters explained that, if the recovery and transfer positions are adjusted correctly, most molds can be filled with a single injection speed. Family molds, with cavities of different volumes and distinct geometries, may require more than one speed. Now, most molds can be filled with a single injection speed; do not use a speed profile if you don't need it.

- b) Adjust the pressure limit equal to P_{max} .
- c) Using velocity V_{95} , adjust the recovery position so that the mold (including parts and runner) fills to approximately 95% of its total weight.

At first glance, the parts may appear completely filled, since molten thermoplastics are compressible and will expand to occupy a significant portion of the volume; even so, it is still missing material.

Even when the velocity is reduced by 5%, it's possible to reach the set pressure limit, since P_{Max} was found with incomplete filling (around 80%). Always verify that the pressure limit is never reached; keep it 5% to 10% above the required pressure.

Some molds present significant filling challenges. For instance, in micro-molding or with long thin parts (thin wall values exceeding 250), it's necessary to fill over 95% during the injection stage. To accomplish

this in those distant and uncomfortable spaces, it's crucial for the melt to have the lowest possible viscosity and flow rapidly toward the end of fill.

Notes:

- Do not try to maximize the fill during the injection stage; only guarantee about 95%. Trying to get to the maximum can result in problems with flash and screw bounceback.
 - Be sure to produce parts without flash.
 - Ensure that the pressure during injection remains at least 5% lower than the set pressure limit.
- d) Without changing the transfer position and continuing with the hold stage off (hold time = 0 or hold pressure = 0), create a rheology table with different injection speeds.

Remember that this laboratory can be performed:

- With machine rheology by power or by viscosity.
- With complete or approximated rheology.

We **Universal** molders prefer and recommend approximated rheology by power, since the equations represent the evaluated effects.

Include in the header:

- For machine parameters: injection speed, fill time, hydraulic transfer pressure and plastic transfer pressure.
- For rheology by power: flow and power.
- For conventional rheology: shear rate and viscosity.

Remember to write the corresponding units.

Machine Parameters				Rheo. by Power		Rheo. by Viscosity	
Speed (mm/s)	T _{inj} (s)	P _{hydraulic} (bar)	P _{plastic} (bar)	Flow (bar*cc/s)	Power (bar*c/s)	Sh. Rate (1/s)	Viscosity (bar*s)

VIII-5. Rheology table headers

With rheology by power, use the following equations:

$$\text{Average injection flow} = \frac{\text{injection volume}}{\text{injection time}}$$

$$\text{Injection volume} = \text{screw area } (D^{2\pi/4}) \times \text{injection displacement}$$

$$\text{Peak power} = \text{average injection flow} \times \text{pressure at the transfer position}$$

$$\text{Plastic pressure} = \text{hydraulic pressure} \times \text{intensification ratio}$$

Each of these equations was explained in detail in the “Machine Rheology” chapter.

With approximated power rheology, perform the laboratory using the first and last data points. The initial velocity will be equal to 95% of the maximum velocity found in the previous lab ($V_{95} = 0.95 \times V_{max}$).

The second and last speed will be equal to 10% of the V_{95} speed.

$$= 0.1 \times V_{95}$$

Speed (mm/s)	T_{inj} (s)	$P_{hydraulic}$ (bar)	$P_{plastic}$ (bar)	Flow (cc/s)	Power (bar*cc/s)
V_{95}					
$0.1V_{95}$					

VIII-6. Table of approximated rheology by power

If you work with complete rheology, calculate and enter 10 descending speed values, the first equal to the V_{95} speed, and the next 9 in $0.1 \times V_{95}$ decrements.

Rheology by Power					
Speed (mm/s)	T_{inj} (s)	$P_{hydraulic}$ (bar)	$P_{plastic}$ (bar)	Flow (cc/s)	Power (bar*cc/s)
V_{95}					
0.9 x V_{95}					
0.8 x V_{95}					
0.7 x V_{95}					
0.6 x V_{95}					
0.5 x V_{95}					
0.4 x V_{95}					
0.3 x V_{95}					
0.2 x V_{95}					
0.1 x V_{95}					

VIII-7. Table showing conventional rheology by power

- e) Without turning on the hold stage and without changing the transfer position, inject at the different injection speeds, taking readings of:
 - Maximum hydraulic pressure: Take the reading in the position where the injection ends. If the machine gives plastic pressure readings, then eliminate or leave the hydraulic pressure spaces blank.
 - Fill time: Take the reading of the time it takes from the start of injection to the transfer position.
- f) Calculate and fill the table with its corresponding values. Remember that the maximum plastic pressure = $P_{hydraulic} \times R_i$. If the machine gives a plastic pressure reading, ignore this calculation or assume that $R_i = 1$.

Summary of the steps:

- a) Adjust the injection speed or speeds to 95% of the maximum speed (V_{max}) that was found and call it V_{95} :

$$V_{95} = 0.95 \times V_{max}$$

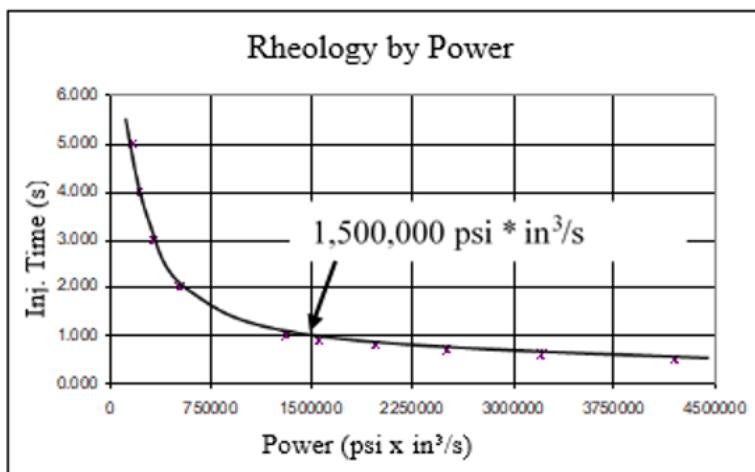
- b) Adjust the pressure limit equal to P_{max} .
- c) Using the V_{95} speed, adjust the recovery position so that it fills the mold (parts and runner) to about 95% of its total weight. Ensure that the

- adjusted pressure limit is at least 5% higher than the maximum injection pressure.
- d) Without changing the transfer position and continuing with the hold stage off, create a rheology chart at different injection speeds.
 - e) With the hold stage still turned off and without changing the transfer position, inject using the different injection speeds, taking readings of:
 - Maximum hydraulic pressure: Take the reading in the position where injection ends. If the machine gives plastic pressure readings, then eliminate these spaces or leave them blank.
 - Fill time: Take the reading of the time it takes from the start of injection to the transfer position.
 - f) Calculate and fill the table with its corresponding values. Remember that the maximum plastic pressure = $P_{hydraulic} \times R_i$. If the machine gives a plastic pressure reading, ignore this calculation or assume that $R_i = 1$.

Laboratory III - Rheology Graph and Determination of Ideal Injection Time and Speed

Using the rheology table already completed, make a rheology graph. If you work with rheology by power, your rheological table will look something like this:

Rheology by Power, $R_i = 6.41$, Vol = 273 in ³					
Speed (in/s)	T_{inj} (s)	$P_{hydraulic}$ (psi)	$P_{plastic}$ (psi)	Flow (in ³ /s)	Power (psi*in ³ /s)
3.2	0.50	1200	7692	546	4199832
2.9	0.60	1100	7051	455	3208205
2.6	0.70	1000	6410	390	2499900
2.3	0.80	900	5769	341	1968671
1.9	0.90	800	5128	303	1555493
1.6	1.00	750	4808	273	1312448
1.3	2.00	600	3846	137	524979
1.0	3.00	550	3526	91	320821
0.6	4.00	500	3205	68	218741
0.3	5.00	461	2955	55	161344



VIII-8. Example of a table and graph of conventional rheology by power

According to this graph, the power stops contributing significantly to the injection time after a power greater than 1,500,000 psi * in³/s.

Laboratory IV - Approximated Graph

As explained in the previous chapter, developing an injection molding rheology laboratory consumes time and resources. With approximated rheology, a mathematical prediction technique, the laboratory can be performed in less than a third of the time.

In the previous example, we used velocity V_{95} and 10% of V_{95} to find the readings for injection time and pressure at the moment of transfer. If we work with rheology by power, you will notice that there is a relatively linear relationship between average injection flow and peak power of injection. Using the equation of a line, $Y = Y_o + MX$, where Y is the peak power, Y_o is the intercept in the coordinate of the peak power, M is the slope, and X is the average flow.

The value of these constants can be found with simple math or Excel.

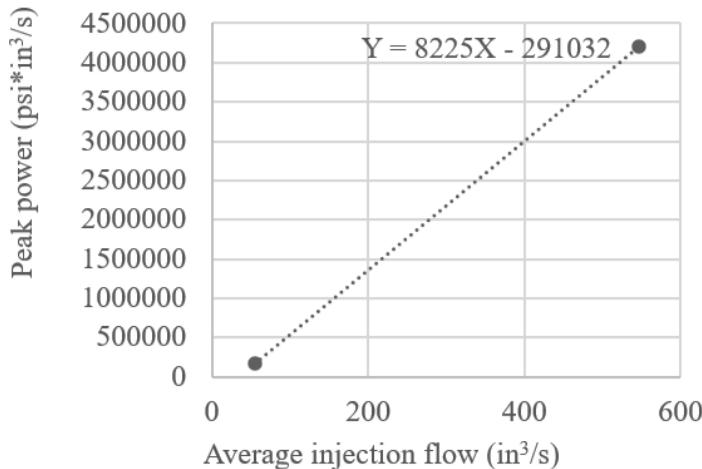
Using the values of the fastest velocity and the slowest velocity from the previous example, we get:

Rheology by Power, $R_i \times 6.41$, Vol 273 in ³					
Speed (in/s)	T_{inj} (s)	$P_{hydraulic}$ (psi)	$P_{plastic}$ (psi)	Flow (in ³ /s)	Power (psi*in ³ /s)
3.2	0.50	1200	7692	546	4199832
0.3	5.00	461	2955	55	161344

VIII-9. Example of a table of approximated rheology by power

Using the Graphing Trendline tool in Excel, we find the linear equation between the maximum and minimum points.

Linear Effect between Peak Power and Average Flow



VIII-10. Graph with linear equation between peak power and average injection flow

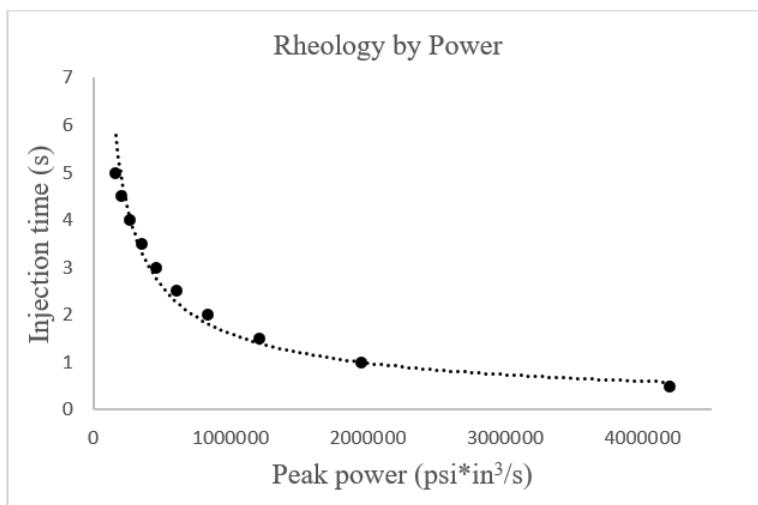
$$\text{Peak power} = 8225 \times (\text{average flow}) - 291032$$

Knowing that the injection volume required is 273 in³ and using the equation for average flow (required volume/injection time), we get:

$$\text{Average flow} = 273 \text{ in}^3 / \text{injection time}$$

With these two equations, we can now approximate the intermediate values of the graph. With ten equidistant injection times, between the minimum and maximum injection times, we calculate their corresponding average injection flow and peak injection power.

$T_{\text{injection}}$ (s)	Flow (in ³ /s)	Power (psi*in ³ /s)
0.5	546.0	4199832
1	273.0	1954393
1.5	182.0	1205918
2	136.5	831681
2.5	109.2	607138
3	91.0	457443
3.5	78.0	350518
4	68.3	270324
4.5	60.7	207951
5	55	161344



VIII-11. Table and graph of approximated rheology by power

Laboratory V. Injection Time Prediction

In the chapter on machine rheology, it was established that the ideal injection time would depend on the type of industry.

Conventional molding industries, where much of the industry is found:

$$T_{\text{Intermediate}} = T_{\min} + (T_{\max} - T_{\min}) / 18$$

Industries that mold sensitive materials, such as rigid PVC:

$$T_{\text{sensitive mat.}} = T_{\min} + (T_{\max} - T_{\min}) / 12$$

Industries with a high volume of injection, such as cap molding and micro-molding:

$$T_{\text{fast}} = T_{\min} + (T_{\max} - T_{\min}) / 36$$

In addition, T_{plateau} was defined as a time when the contribution of power to the injection time begins to be insignificant.

$$T_{\text{plateau}} = T_{\min} + (T_{\max} - T_{\min}) / 9$$

Where:

T_{\min} = injection time for the maximum injection speed
 T_{\max} = injection time for the minimum injection speed

Using the previous example, we find that T_{\min} is 0.5 seconds, which corresponds to the speed of 3.2 in/s (V_{95}) and T_{\max} is 5.0 seconds, which corresponds to the speed of 0.3 in/s (10% of V_{95}).

Rheology by Power, $R_i \times 6.41$, Vol 273 in³					
Speed (in/s)	T_{inj} (s)	$P_{\text{hydraulic}}$ (psi)	P_{plastic} (psi)	Flow (in³/s)	Power (psi*in³/s)
3.2	0.50	1200	7692	546	4199832
0.3	5.00	461	2955	55	161344

VIII-12. Table of approximated rheology by power

- a) After identifying the industry type, select its corresponding equation, and replace the values of T_{\min} and T_{\max} , in order to determine the ideal injection time.

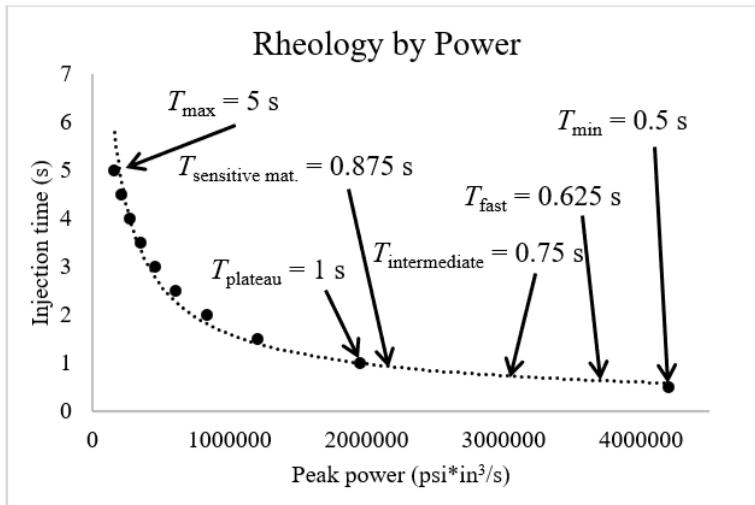
$$\begin{aligned} T_{\text{Intermediate}} &= T_{\min} + (T_{\max} - T_{\min}) / 18 \\ &= 0.5\text{s} + (5\text{s} - 0.5\text{s}) / 18 = 0.75 \text{ seconds} \end{aligned}$$

$$T_{\text{Sensitive mat.}} = T_{\min} + (T_{\max} - T_{\min}) / 12$$

$$= 0.5\text{s} + (5\text{s} - 0.5\text{s}) / 12 = 0.875 \text{ seconds}$$

$$\begin{aligned} T_{fast} &= T_{min} + (T_{max} - T_{min}) / 36 \\ &= 0.5\text{s} + (5\text{s} - 0.5\text{s}) / 36 = 0.625 \text{ seconds} \end{aligned}$$

$$\begin{aligned} T_{plateau} &= T_{min} + (T_{max} - T_{min}) / 9 \\ &= 0.5\text{s} + (5\text{s} - 0.5\text{s}) / 9 = 1.0 \text{ seconds} \end{aligned}$$



VIII-13. Rheology graph indicating injection times by industry type

These equations help standardize the selection of injection time; even so, they are only a reference.

c) Determine the ideal injection speed.

Once you have determined the injection time, find the corresponding injection speed. This is done by adjusting the injection speed until it approximately equals the ideal injection time found.

With the ideal speed found, verify that the mold (parts and runner) fills **close to 95%** of its total weight or volume. If not, adjust the recovery position until both the ideal injection time and a fill of about 95% are achieved. Keep in mind that once the pack stage is optimized, the final

position could come close to zero if the injection volume is significantly below 95%.

Remember that not every mold can be filled to around 95% during the injection stage; some because of a condition of their design and in some cases, because the mold must be repaired. For example, in molds with filling difficulties, such as in micro-molding or long and thin parts, it is necessary to achieve over 95% filling during the injection stage. To accomplish this, in those distant and uncomfortable spaces, it is crucial for the molten material to have the lowest possible viscosity and flow rapidly toward the end of fill.

Injection speed may cause defects. There are several scenarios:

- Burns at the end of fill - the vents are likely to be plugged or defective.
- Burns around edges or corners - could be caused by an edge or flash of metal in the mold.
- Burns in the form of streaks that extend along the fill or from the gate - it is likely that the material is degrading by friction, etc.

Consult the mold maintenance department; it will probably recommend some type of maintenance or repair. Follow the recommendations and return to molding at the determined injection speed.

Unfortunately, not every material, mold, and molding equipment is at its best design condition. What is worse, for whatever reason, is being forced to mold with these deficiencies.

In the event of not being able to do the rheology as a result of material limitations (e.g. degradation) or equipment limitation (e.g. inability to rapidly inject), if parts must be molded under these conditions, follow this simple procedure:

- Inject by incrementing the speed until it reaches the specific limitation.
- Then reduce the maximum speed found by 5%, and inject with this new speed; if the defect disappears this will be your injection speed or time.

d) Note the values found for:

- injection time
- injection speed and its corresponding injection flow
- transfer position and its corresponding transfer volume
- plastic pressure limit and hydraulic pressure limit (if applicable)
- recovery position and its corresponding recovery volume.

Summary of the steps:

- a) Using the completed rheology table, create the rheology graph.
- b) Select the ideal injection time for your application.
- c) Select the ideal injection speed corresponding to the injection time found.

d) Note the values found for:

- injection time
- corresponding injection flow
- corresponding transfer volume
- plastic pressure limit
- corresponding recovery volume

Notes:

- Do not try to maximize the fill, just guarantee something close to 95%. Trying to get to the maximum can result in flash problems.
- Be sure to produce parts without flash.
- Ensure that pressure during injection is maintained at least 5% lower than the set pressure limit.
- There is a relatively linear relationship between injection flow and peak power.
- Remember that you are working with expensive equipment: do not rush the work and follow the safety rules established by your factory and government agencies.

Questions

- 1) During a ***Universal MoldingTM*** lab, what do you do if some limitation or defect in equipment appears, such as limited injection speed?
 - a. If the limitation requires a modification or a simple change of equipment, do it. If the modification or change is not feasible, or is economically unsustainable, you will have to run the laboratory with the equipment you have.
 - b. If the limitation requires a modification, do it. The laboratory must be carried out under ideal conditions.
- 2) All programmed injection speeds, whether slow or fast, are always reached by the injection unit.
 - a. True.
 - b. False, the speed entered may be limited by an inadequate injection unit.
- 3) While determining the injection speed, hold pressure should be set to an average pressure.
 - a. True, hold is used during the determination of injection time.
 - b. False, hold is turned off so that it does not interfere with the determination of injection speed.
- 4) While determining the minimum injection time, the injection unit should be adjusted to produce parts that are 20% incomplete.
 - a. True, this is done to prevent damage to the mold, the machine, or both.
 - b. False, it is necessary to fill the total volume.
- 5) With fill of incomplete parts, for example to 80%, you will notice that the weight of the parts decreases with an increase in injection speed.
 - a. True.
 - b. False, the weight of the parts increases with an increase in injection speed.
- 6) Select all the correct statements:

- a. The fastest speed used in the rheology table is 95% of the maximum speed (V_{max}) found during the determination of the minimum injection time.
- b. The rheology table is created by injecting at different speeds, all with an incomplete fill of 80%.
- c. Ensure that the pressure during injection is maintained at least 5% lower than the set pressure limit.

7) Select the correct statement:

- a. The power rheology graph coordinates are viscosity versus changing speed.
- b. The power rheology graph coordinates are injection time versus peak power.
- c. The conventional rheology graph coordinates are fill flow versus peak power.

8) Select all the correct statements:

- a. There is a relatively linear relationship between viscosity and injection flow.
- b. There is a relatively linear relationship between injection flow and power.
- c. With approximated rheology by power we use the equation of a line, $Y = Y_o + MX$, where Y is the flow, Y_o is the intercept in the injection flow coordinate, M is the slope and X is the injection's peak power.

9) Materials sensitive to injection speed, such as rigid PVC, need an injection time

- a. between $T_{plateau}$ and T_{min} .
- b. close to T_{min} .
- c. close to $T_{plateau}$.

10) Once you have determined the ideal injection time, find the corresponding injection speed.

- a. Also, verify that the mold (parts and runner) fills to about 80% of its total weight.
- b. Hold continues to be turned off until this part of the lab.
- c. Ensure that the pressure during injection is maintained to at least 5% lower than the set pressure limit.
- d. All the above are correct.

IX. Verifying Fill Balance

- **Effect of Injection Time on Fill Balance**
- **Laboratory - Fill Balance**
- **Thermal Imbalance**
- **Unbalanced Fill in Molds with Cold Runners**

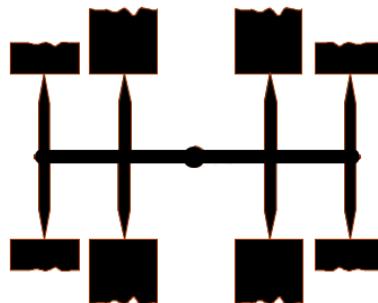
Again, before proceeding with this ***Universal Molding™*** laboratory remember that:

- Auxiliary equipment should be properly installed and correctly operating.
- Temperatures should already be reached.
- Barrel adjustments should have been programmed.
- The required clamping force should have been set.
- The press openings, their movement, speeds, and mold protection should have been carefully and precisely adjusted.
- Extended cooling time should have been set, in order to prevent it from interfering in the determination of other parameters.
- The ideal injection speed should have been determined, adjusted and should be filling about 95% of the required fill for the mold.
- The injection pressure limit should have been determined and adjusted.
- The hold stage should stay off.

Important -- only qualified personnel who have read the operational manuals of the equipment and understand the functionality of the equipment should operate and/or adjust them.

Multi-cavity molds increase productivity. However, their fabrication, as well as their operation, is complicated; the melt's flow path must be balanced, and cooling is more elaborate.

Imagine a multi-cavity mold with distinct fill times for each cavity, or with some cavities that fill before others.

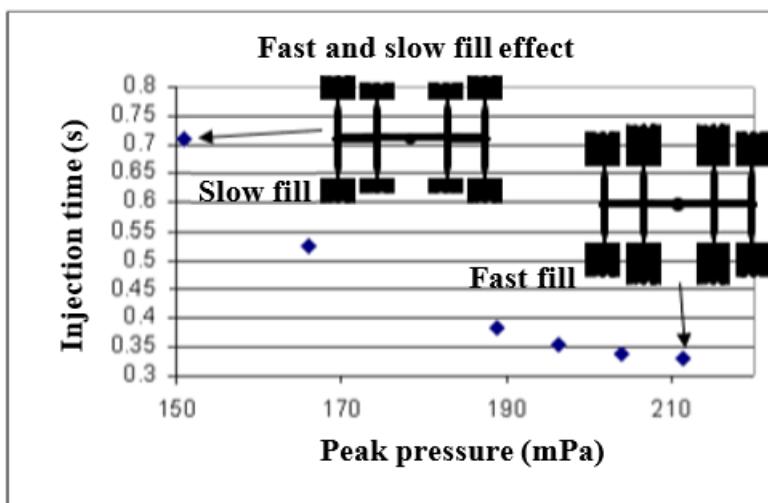


IX-1. Multi-cavity mold with unbalanced fill

This situation defeats the objective of maintaining constant flow and viscosity per cavity; it is like accepting the molding of distinct parts for each cavity.

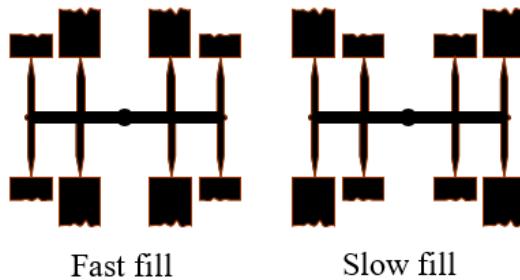
Effect of Injection Time on Fill Balance

The "Determining Injection Speed" chapter mentioned the multiple effects of injection time on fill. In the injection stage, the volume of melt in the parts increases with an increase in injection speed. Look at the illustration.



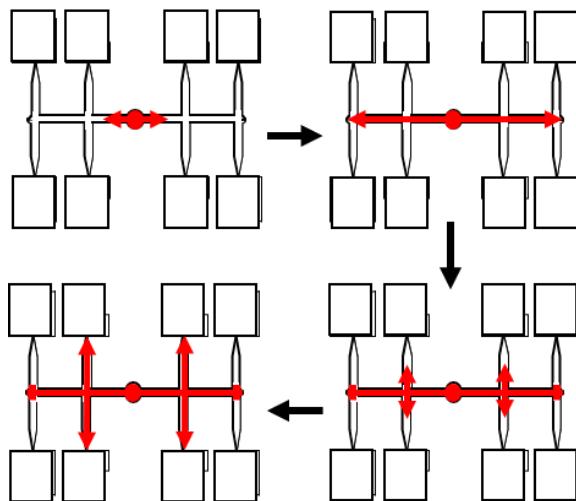
IX-2. Graph of the effect of injection speed on the fill of incomplete parts

In addition, it can be noted that with a fast fill, the interior cavities will be filled sooner than the exterior ones.



IX-3. Fast and slow fill

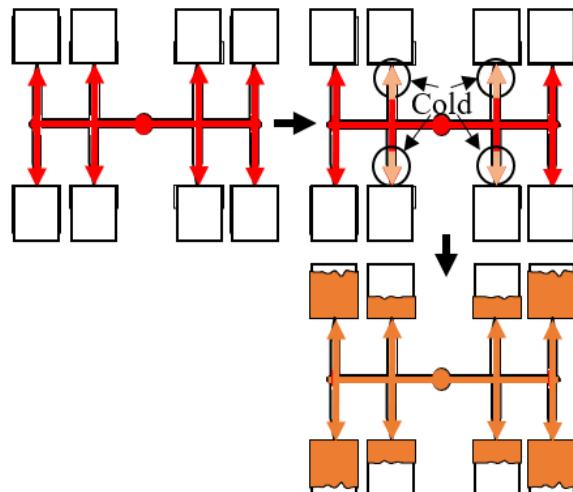
Let's see why; during the start of fill, the flow is divided between both sides of the sprue. Melt flow will always prefer the least restriction, so it will continue in a straight line and not turn into the first intersections where the runner branches out. Once it reaches the end of the main branch and is forced to turn, it prefers filling the previous intersection, since the melt pressure will be greater at that intersection.



IX-4. Example of flow sequence of the runner

Now, once the melt reaches the first gates, it encounters the restriction of those narrow openings. Then, the melt will prefer to continue flowing through the runner, since it is the path of least restriction, until it encounters the same difficulty, the restriction of small gates at the end.

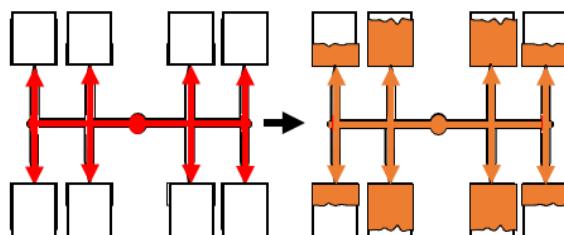
Let's assume the fill is slow. The melt in these outer gates is still hot and has less viscosity than the melt in the inner gates, as it has been in contact with the cold metal for longer.



IX-5. Example of slow fill

Consequently, the exterior cavities will be filled first. During injection, the melt in contact with the cold metal is the first to harden, and the thickness of this hardened layer will depend on the amount of time the melt is in contact with that cold metal. When the injection is slow, the hardened layer is thicker and, as a result, the melt path is narrower.

Now let's do the same exercise, but this time do it assuming the flow is fast. Initially the runner will fill in the same manner as before. The difference is that the time that the melt sits in the first runners is less.



IX-6. Example of fast fill

Consequently, its viscosity will stay low and the melt will prefer to fill the interior cavities, since the required pressure will be less.

Injection time has a lot to do with the flow balance, which is why the injection speed must be determined and adjusted before carrying out a study of flow balance.

Other situations caused by imbalance:

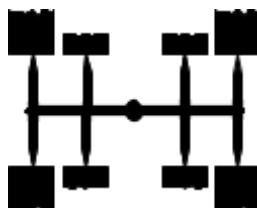
1. Jamming or difficulty in ejection:
 - Some cavities may cause parts to get stuck or exhibit ejection difficulties. These cavities probably fill first.
 - On the other hand, parts that detach on their own (with minimal or no effort from ejectors) are probably the last to fill.
2. Flash formation:
 - Cavities that frequently show flash (excess material) are most likely the ones that fill first.
3. Higher part weight:
 - Parts with greater weight also have a probability of filling first.

Understanding these scenarios is essential, as they can impact the overall molding process.

Laboratory - Fill Balance

This section will present the procedure for verifying flow imbalance:

1. Adjust the injection unit to produce incomplete parts and make sure that hold is still turned off.

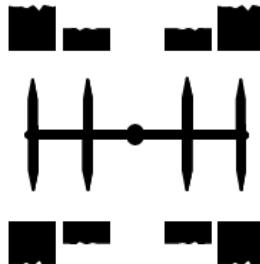


IX-7. Example of incomplete fill

This experiment is carried out with incomplete fill so that we can evaluate the fill of each individual cavity. If one or more cavity completely fills, it

would be impossible to determine the fill balance. Try to achieve a fill of 80%, then verify that each cavity is incomplete; if any is not, reduce the recovery volume even more and reverify.

2. If the parts are attached to the runner, separate them.



IX-8. Parts separated from the runner

3. Weigh the parts separately and identify them with W_i ($i = 1$ to # of cavities). Add all the W_i weights and call this W_T .

$$W_T = \sum_{\# cav.}^1 W_i$$

4. Determine the volume deviation for each cavity, Vd_i .

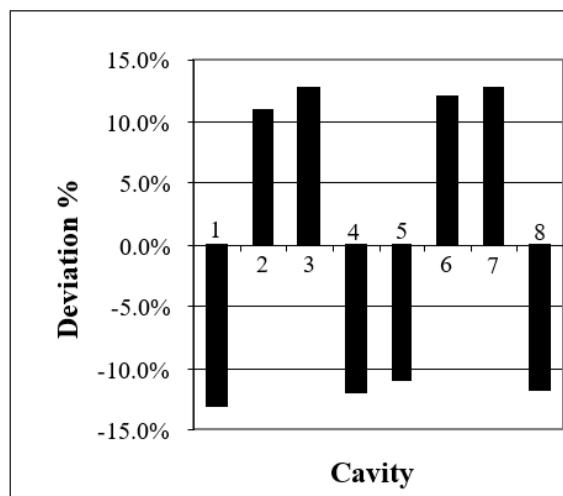
$$Vd_i = \left[\frac{W_i}{\left(\frac{W_T}{\# cavities} \right)} - 1 \right] 100\%, \quad i = 1 \text{ to } \# \text{ of cavities$$

If the determined percentage is zero, the flow is ideal; if it is positive, the flow is higher than ideal; if it is negative it is lower than ideal. You should normally try to maintain this percentage to within $\pm 10\%$. Verify the deviation that is permitted by your company. Example:

Cavity	Sample 1	Sample 2	Average	Deviation
1	3.42	3.42	3.42	-13.1%
2	4.36	4.37	4.365	10.9%
3	4.42	4.44	4.43	12.5%
4	3.46	3.47	3.465	-12.0%
5	3.5	3.5	3.5	-11.1%
6	4.4	4.41	4.405	11.9%
7	4.43	4.44	4.435	12.7%
8	3.47	3.47	3.47	-11.8%

IX-9. Table with incomplete cavity weights and corresponding fill deviations

The data demonstrates a negative deviation in cavities 1, 4, 5, and 8, indicating that they are the last ones to be filled, and a positive deviation for cavities 2, 3, 6, and 7, indicating that they are the first to be filled. Look at the column graph with the results.



IX-10. Column plot with % of cavity fill deviation

Thermal Imbalance

It is good to have and understand the mold cooling water flow diagrams. Cooler cavities tend to harden plastic faster and, consequently, make the flow more viscous.

Some companies use infrared systems to determine their molds' thermal distribution. Remember that the temperature of metal, plastic, and water are usually distinct, although they are still related.

In the case of hot runners, the flow can be corrected by zone, by changing the temperature at the hot drops of the runner system.

For example:

- In the case of a negative Vd_i , the temperature of that hot drop should be increased. Be careful not to overheat the plastic in the drops.
- In the opposite case where the Vd_i value is positive, the temperature of the hot drop should be lowered. Be careful not to cause the plastic to solidify in the drops.

It is crucial to understand that when you modify the flow in one zone, the others will also be affected. For this reason, it is recommended to make one change at a time and inject new parts. Additionally, consider starting by cooling the zones of the cavities that fill first, which will yield faster results.

It is important to understand that once the flow of one zone is changed, the other zones will be affected. Therefore, it is recommended making one change at a time and injecting new incomplete parts.

This procedure works with hot runner systems with valve gates. Just make sure the valves remain open during the injection stage.

When the imbalance is such that it cannot be thermally corrected, the correction is made on the tooling.

Imbalance in Molds with Cold Runners

In the case of cold runners, the correction is not so simple. The imbalance could be a result of the cavity's water flow and temperature by cavity, it could be that the water passages are dirty, or it could be a mold design problem.

For example:

- Cavities with a positive Vd_i value may be receiving less cooling. Possibly there is an obstruction in the water passages.
- Cavities with a negative Vd_i value may be receiving more heat removal. It could be that the water passages are incorrectly connected.

Runners that distribute melt to cavities with a negative Vd_i could be enlarged to facilitate the passage to these cavities. Important! This practice should only be performed by experienced and knowledgeable tooling technicians who know the proper techniques.

Questions

- 1) Before verifying the fill balance,
 - a. adjust the injection speed to V_{95} , the recovery to completely fill the mold, and the fill to a minimum pressure.
 - b. adjust the speed to 80% of what's required, the pressure limit to what has been determined, and the fill to an average pressure.
 - c. the ideal injection speed and injection pressure limit should be determined and adjusted, and the hold stage should stay off.
- 2) Choose the correct statement:
 - a. Verification of the fill balance is performed with a fill of slightly more than 95% of the mold.
 - b. Verification of the fill balance is carried out with an incomplete fill, for example, 20% incomplete.
 - c. Verification of the fill balance is performed after the transfer position has been adjusted to a minimum hold.
- 3) The injection time has a lot to do with the flow balance, which is why the injection speed must be determined and adjusted before doing a study of the flow balance.
 - a. True
 - b. False
- 4) In an analysis of flow balance, if the percentage determined by the volume deviation equation is zero, the flow is ideal.
$$(Vd_i = \left[\frac{W_i}{\left(\frac{W_T}{\# cavities} \right)} - 1 \right] 100\%, i = 1 \text{ to } \# \text{ of cavities})$$
 - a. True
 - b. False
- 5) In an analysis of flow balance, if the percentage determined by the volume deviation equation is
 - a. positive, the flow is above ideal.
 - b. negative, the flow is below ideal.

- c. zero, the flow is ideal.
- d. all the above.

6) In an analysis of flow balance, the following was obtained:

Cavity	Sample 1	Sample 2	Average	Deviation
1	3.42	3.42	3.42	-13.1%
2	4.36	4.37	4.365	10.9%
3	4.43	4.44	4.435	12.7%
4	3.47	3.47	3.47	-11.8%

- a. The result shows a negative deviation in cavities 1 and 4, indicating that they are the first to fill.
- b. The result shows a positive deviation in cavities 2 and 3, indicating that they are the first to fill.

7) In an analysis of flow balance with hot runners,

- a. we can compensate the flow by zone by changing the temperature in the hot drops of the runners.
 - b. in the case where the volume deviation is negative, the temperature of the hot drop of that cavity should be lowered.
 - c. in the case where the volume deviation is positive, the temperature of the hot drop of that cavity should be increased.
- 8) Correcting the flow balance in molds with cold runners is not so simple, since controlling the water flow and its temperatures could be a mold design problem.
- a. Runners and gates that distribute the melt to the cavities with a positive volume deviation should be enlarged to facilitate passage to these cavities.
 - b. Runners that distribute the melt to the cavities with a negative volume deviation may need to be enlarged to facilitate passage to these cavities, carried out only by experienced and technically knowledgeable tooling technicians.

X. Determining Hold Stage Parameters

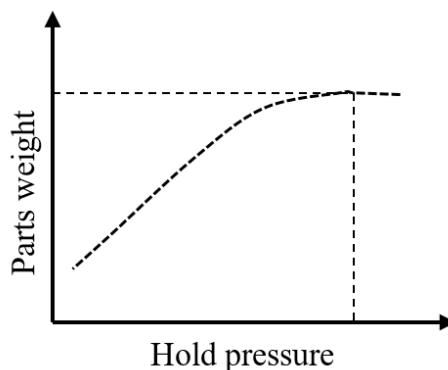
- **Laboratory I - Determining Hold Pressure**
- **Laboratory II - Determining Hold Time**
- **Hold Stage for Molds with Hot Runners and Gate Valves**

Again, remember that before continuing with these ***Universal MoldingTM*** labs:

- Auxiliary equipment should be properly installed and correctly operating.
- Temperatures should already be reached.
- Barrel adjustments should have been programmed.
- The required clamping force should have been set.
- The press openings, their movement, speeds, and mold protection should have been carefully and precisely adjusted.
- Extended cooling time should have been set, in order to prevent it from interfering in the determination of other parameters.
- The ideal injection speed should have been determined, adjusted and should be filling about 95% of the required fill for the mold.
- The injection pressure limit should have been determined and adjusted.
- The hold stage should stay off.
- The fill balance should have been verified and adjusted.

Important -- only qualified personnel who have read the operational manuals of the equipment and understand the functionality of the equipment should operate them and/or make adjustments.

The hold pressure is determined, increasing the hydraulic or melt hold pressure, until the cavities are completely full or have reached the desired weight.



X-1. Graph of the effect of hold pressure on the weight of the parts

Notes:

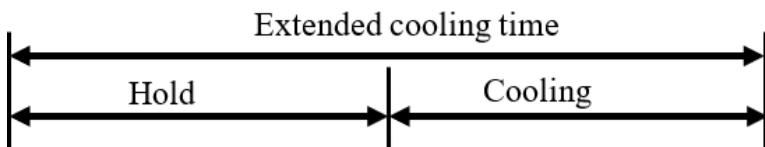
In this laboratory, only the pieces without runners are weighed.

There exists parts that cannot be weighed, such as overmolded terminals with long cables and parts that must be demolded together with the runner. If it is not possible to weigh the parts, consider the following:

- Some dimension, such as the thickness of a wall in one or multiple cavities: If the pack pressure increases, the thickness of that wall will also increase.
- Cushion position or minimum screw position after holding: If the holding pressure increases, the cushion will decrease.

Laboratory I - Determining Hold Pressure

1. Set the hold time to a value greater than required. For example, consider half the extended cooling time that was used in previous labs. The goal is to ensure gate freeze; hold time will be optimized later.
2. Write down a cooling time equal to the difference between extended cooling time and hold time. Note that the sum between hold time and cooling time will be equal to extended cooling time.



X-2. Extended cooling time

For example, in a laboratory where the extended cooling time is equal to 12 seconds, half of the extended cooling time was used in order to guarantee gate freeze. The results are the following:

$$\begin{aligned} \text{Hold time} &= 6 \text{ seconds} \\ \text{Cooling time} &= 6 \text{ seconds} \end{aligned}$$

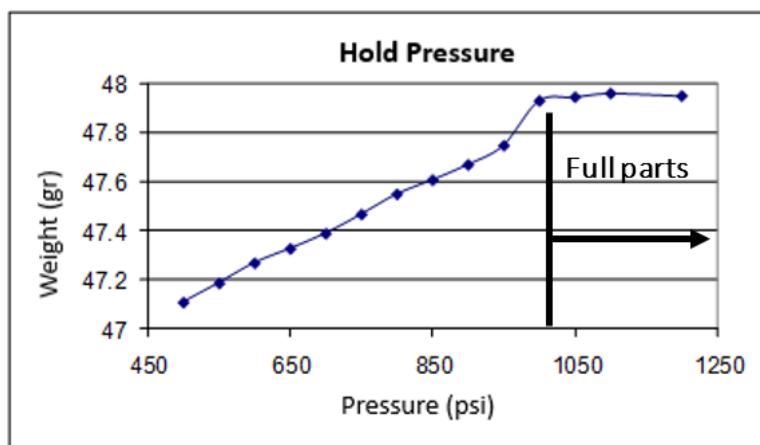
Important, always guarantee that cooling time is longer than recovery time. It was explained in previous chapters that, when cooling time expires, permission to open the mold will only be allowed if recovery has finished. In other words, the mold will be closed until the recovery is complete, regardless of whether cooling time has finished. Consequently, cooling time will be extended and, depending on the machine, you may not realize this is happening.

3. Perform a weight study of the parts at different hold pressures. Start the experiment with a lower pressure than required. For example, start with 10% of the injection pressure acquired at the time of transfer; then increase the pressure until you get completely full parts or the desired weight. Take two or three samples per pressure and calculate the average of the weight of the parts. Tabulate the data and plot the weight of parts against hold pressure. Weigh only the molded parts; do not include the runner.

Pressure (psi)	Average weight (gr)
500	47.11
550	47.19
600	47.27
650	47.33
700	47.39
750	47.47
800	47.55
850	47.61
900	47.67
950	47.75
1000	47.93
1050	47.95
1100	47.96
1200	47.95

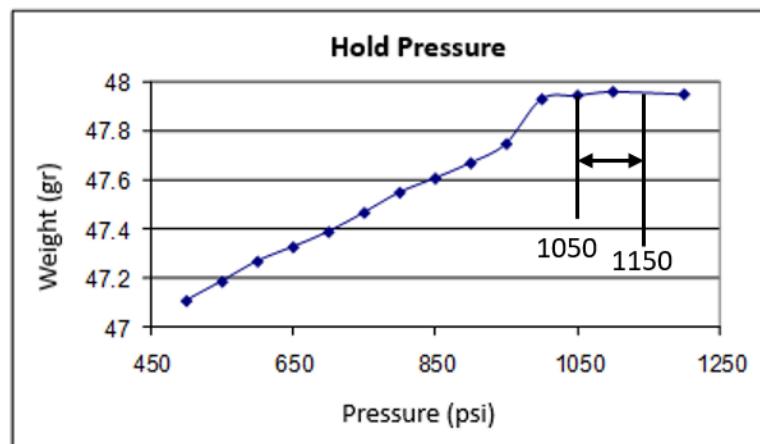
X-3. Table of the effect of hold pressure on the weight of the parts

In the graph below, we see that the pressure stopped contributing to the weight of the parts after 1000 psi of hold pressure.



X-4. Graph of the effect of hold pressure on the weight of the parts

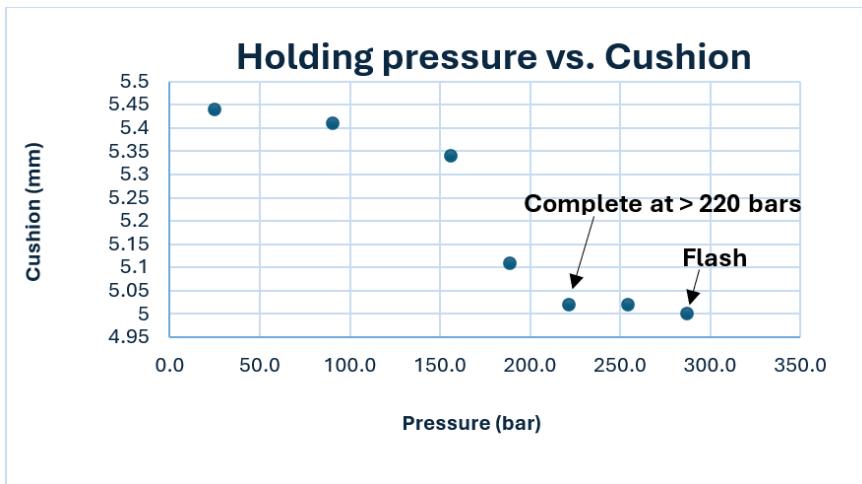
4. Select a pressure where the weight is relatively constant. This example used a minimum hold pressure of 1050 hydraulic psi, a machine parameter. The norm would be to select a pressure range, for example from 1050 to 1150 psi.



X-5. Graph indicating hold pressure range

- Once you select your hold pressure range, convert to *Universal* parameters.

Example:



In a process of overmolding, where the pieces could not be weighed, it was decided to use the effect of hold pressure on the cushion position (See the graph below). The graph reveals that the cushion stopped contributing to the fill with a pack pressure greater than 220 bars and also showed that pressures greater than 260 bars created flash.

X-6. Determination of hold pressure with the cushion's position

- After selecting the hold pressure, it is essential to verify the percentage of volume injected during the injection stage, based on position. Although the weight-based percentage should have been verified during the determination of the ideal injection time, it is recommended to confirm it using a second method: by position. This is a simple and effective verification method that all injection processors should utilize.

Verification of injected volume percentage based on the screw's position

The formula to calculate this percentage is as follows:

$$\% \text{ volume at injection stage} = \frac{\text{recovery position} - \text{transfer position}}{\text{recovery position} - \text{cushion position}} * 100\%$$

If you are using a control system with **Universal** parameters and working with volume instead of position, apply this formula:

$$\% \text{ volume at injection stage} = \frac{\text{recovery volume} - \text{transfer volume}}{\text{recovery volume} - \text{cushion volume}} * 100\%$$

Example:

Determine the fill percentage during the hold stage in a process where the recovery position is $PP = 32\text{mm}$, the transfer position is $PT = 6\text{mm}$, and the cushion position is $PC = 4\text{mm}$.

$$\% \text{ volume at hold stage} = \frac{(PT-PC) \times 100\%}{(PP-PC)} = \frac{(6\text{mm}-4\text{mm}) \times 100\%}{(32\text{mm}-4\text{mm})} = 7\%$$

This equation can be stated more simply:

$$\begin{aligned}\% \text{ volume at hold stage} &= \\ 100\% - \% \text{ volume at injection stage} &\end{aligned}$$

Subsequently, if necessary, make adjustments by incrementally increasing or decreasing the recovery position. Keep in mind that modifying this position will likely impact both injection time and maximum injection pressure. If the adjustment is significant, modify the injection speed to coincide with the previously determined ideal injection time and adjust the injection pressure limit to be 5% to 10% above the new maximum injection pressure.

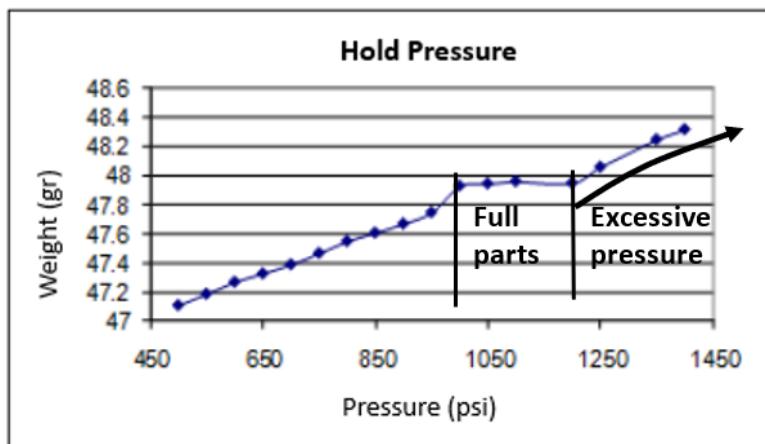
Finally, update your technical documentation with the new recovery and cushion positions, the new maximum injection pressure and injection limit pressure, and the adjusted ideal injection speed.

In this example the injection unit used had an intensification ratio (R_i) of 12.2, and its corresponding **Universal** parameters are:

Universal hold pressure = Hydraulic pressure x R_i
Minimum pressure = 1050 psi x 12.2 = **12810 psi**
Maximum pressure = 1150 psi x 12.2 = **14030 psi**
Average pressure = 1100 psi x 12.2 = **13420 psi**

Notes:

- Although the objective is dimensions and not weight, for now you must work with the weight of the parts. After unmolding, the parts will undergo multiple changes in dimensions due to shrinkage.
- Only weigh the molded parts, not the runner.
- Avoid opening the mold or forcing melt to sneak into unwanted spaces as a result of excessive pressure. An easy way to find out if the pressure is excessive is by looking at the graph of pressure versus weight of the parts.

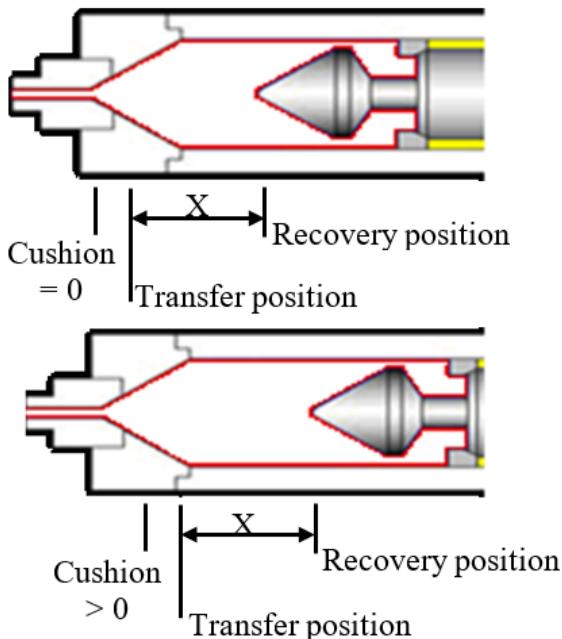


X-7. Graph indicating excessive hold pressure

In the graph, after 1200 psi a new tendency was started, indicating that the cavities are opening, or plastic is oozing.

- The cushion must be greater than zero. If it reaches zero, injection volume will have to be increased. This could have happened as a result of a filling volume well below 95% during the injection stage. Assuming there are no defects in the injection unit, such as a leaky check ring, increase the recovery position and transfer position by the same amount until the final position (after hold) is greater than zero. If you want to make this adjustment without stopping the process, first increase the

cushion position and then the recovery position. Otherwise, you could end up with excess material, which might damage some molds.



X-8. Correction when cushion is zero

If the cushion becomes zero as a result of a defective injection unit, such as a leaky check ring, talk to the maintenance department to have the defect repaired.

Notes:

The final cushion position is always greater than zero and less than the transfer position.

If the cushion position ends up being greater than the transfer position, there are two possible scenarios:

1. Low hold pressures: In some machines, when hold pressures are programmed too low, the cushion position can move above the transfer position. This situation is resolved by programming the appropriate hold pressure.
2. Excessive volume during injection stage: Another common reason is when positions are programmed that result in a filling volume

exceeding 95% during the injection stage. The solution is to reduce that filling percentage during that stage.

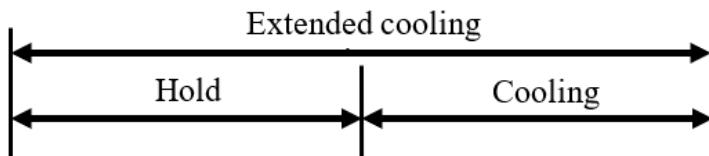
These position adjustments will impact both injection pressure and injection time. Make the necessary adjustments.

Laboratory II - Determining Hold Time

Hold time is determined by the gate freeze test. While the gates are liquid, hold pressure keeps the melt in the cavity. Once the gates freeze, the melt cannot escape even when hold pressure is removed.

1. Adjust to the previously determined hold pressure.
2. Without changing the hold pressure found, decrease the hold time in intervals until the weight of the parts begins to decrease due to soft gates, which are unable to keep the compressed melt inside the cavities.

In order to maintain a constant total cycle, the sum of cooling time and hold time must be kept constant. For each interval that is subtracted from hold time, the same amount should be added to cooling time.



X-9. Extended cooling time

3. Tabulate the times with their corresponding total weights of the parts and make a chart of the weight of the parts against hold time.

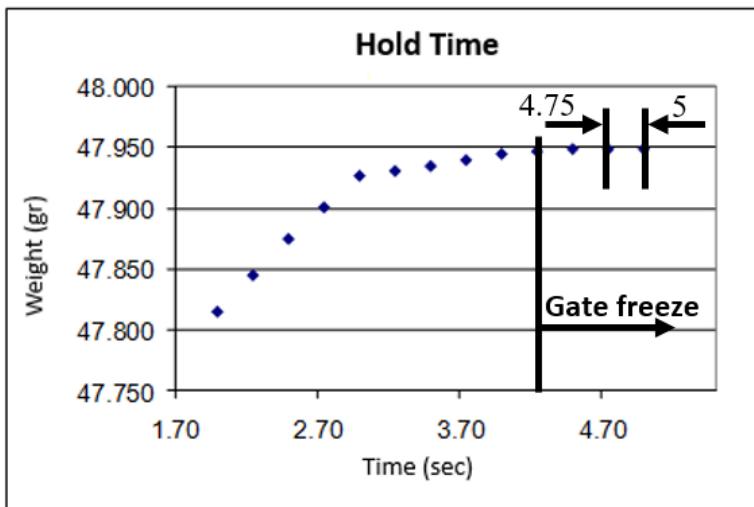
Hold time (s)	Average weight (gr)	Cooling time (s)
5.00	47.949	23.00
4.75	47.949	23.25
4.50	47.949	23.50
4.25	47.947	23.75
4.00	47.945	24.00
3.75	47.940	24.25
3.50	47.935	24.50
3.25	47.931	24.75
3.00	47.927	25.00
2.75	47.901	25.25
2.50	47.875	25.50
2.25	47.845	25.75
2.00	47.815	26.00
1.75	23.908	26.25

X-10. Table of the weight of the parts and their respective hold and cooling times

Although the cooling time column is not graphed, include it. It will help you when entering the times during the experiment.

Notes:

- For each hold time setting, take two to three samples and average the total weight of the parts.
 - Only take the weight of the molded parts without the runner.
 - Always ensure that the cooling time is longer than the recovery time.
4. On the graph, find the time where the weight of the parts begins to decrease. It is clearly shown that hold times longer than 4.5 seconds will be required to ensure gate freeze.



X-11. Graph of the effect of hold time on the weight of the parts

The hold time range selected in this example is 4.75 to 5 seconds.

Summary of parameters

Once the hold parameter determination is complete, summarize them:

- determined hold time and its operational range
- determined hold pressure (**Universal** and machine parameter) and its operational range
- remaining extended cooling time
- volume position or recovery position
- cushion or final position of the screw after hold and its operational range.

Finally, verify that the machine control has been programmed with the determined hold parameters.

Hold Stage for Molds with Hot Runners and Gate Valves

With molds that include hot runners, the procedure is similar; we will have to guarantee gate freeze.

Now, if the mold also includes gate valves, those will do the function of gate freeze. These are programmed so that valve closure occurs when the

adjusted hold time is met. In addition to creating a good finish at the injection site, these valves reduce hold time, since you don't have to wait for gate freeze.

Notes:

- If the valves close prematurely you will probably end up with incomplete parts.
- If the valves close too late, they may encounter hardened material in the valve seat, which could damage the gates in the cavities. Consequently, this defect would be reflected at the injection point of the parts.

During the hold stage, we guarantee measurements that are a function of mass. Do not attempt to correct thermal measurements in the hold laboratory.

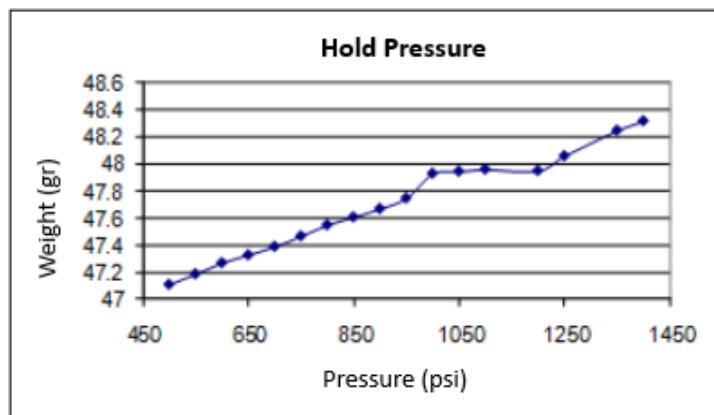
Amorphous	Semi-crystalline
Problems with over-packing due to low shrinkage	Problems of incomplete pack due to high shrinkage
Problems with flash	Sink marks
Breaks during demolding	Easy demolding
The transition from softened melt to solid is gradual	The transition from liquid melt to solid is sudden

X-12. Mechanical characteristics of materials which should be considered during hold

Questions

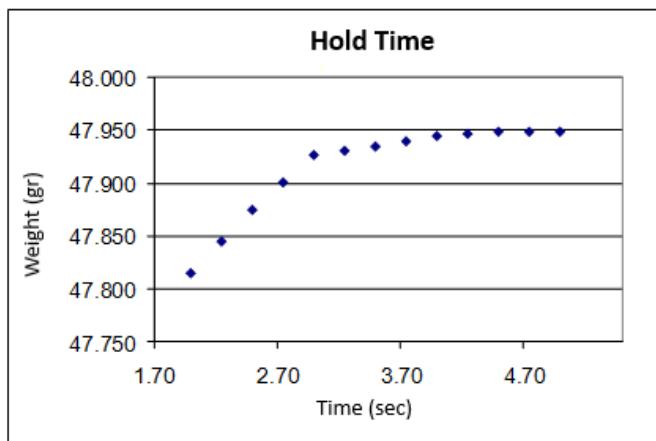
- 1) Increasing hold pressure increases the weight of the parts. Assuming the mold does not open, eventually a pressure will be reached where the weight of the parts will stop increasing.
 - a. True
 - b. False
- 2) A hold pressure graph is created by using hold pressures and the total weights of the parts with runners.
 - a. True, the weight of the parts and runners is used.
 - b. False, only the weight of the parts without the runners is used.
- 3) The dimensions that are the effect of shrinkage are evaluated when determining hold pressure.
 - a. True, during hold we consider the mass dimensions.
 - b. False, during hold we only consider the mass dimensions and not the shrinkage.
- 4) The hold time is used to
 - a. determine thermal dimensions.
 - b. improve the operating cycle.
 - c. determine gate freeze.
- 5) In one process a hold time of 6 seconds was found. The corresponding ***Universal*** hold time would be
 - a. 6 seconds.
 - b. 6 seconds ÷ intensification ratio.
 - c. 6 seconds x intensification ratio.

6) Using the graph below, select all the correct statements.



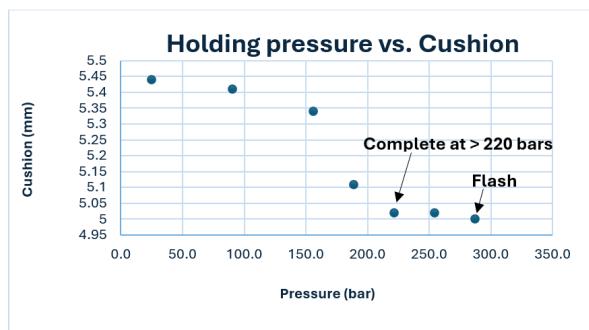
- a. In the graph, after 1200 psi a new tendency begins, indicating that the mold is opening, or plastic is oozing as a result of excessive pressure.
- b. In the graph, using a hold pressure of 850 psi will give us complete parts.
- c. In the graph, a selection of 1050 psi is an adequate hold pressure.

7) Using the illustrated graph, select the correct statement.



- a. We get gate freeze with a hold time of 2 seconds.

- b. We can see that we will need hold times greater than 4.5 seconds in order to guarantee gate freeze.
- 8) Determine the fill percentage during the hold stage in a process where the recovery position is PP , the transfer position is PT , and the cushion position is PC .
- $\% \text{ volume at the hold stage} = \frac{(PT-PC)x100\%}{(PP-PC)}$
 - $\% \text{ volume at the hold stage} = \frac{(PP-PC)x100\%}{(PP-PC)}$
 - $\% \text{ volume at the hold stage} = \frac{(PP-PT)x100\%}{(PP-PC)}$
- 9) Determine the fill percentage during the injection and hold stages in a process where the recovery position is PP , the transfer position is PT , and the cushion position is PC .
- $\% \text{ volume at the injection stage} = \frac{(PP-PC)x100\%}{(PP-PT)}$,
 - $\% \text{ volume at the hold stage} = 100\% - \% \text{ volume at the injection stage}$
 - $\% \text{ volume at the injection stage} = \frac{(PT-PC)x100\%}{(PP-PC)}$
 - $\% \text{ volume at the hold stage} = 100\% - \% \text{ volume at the injection stage}$
 - $\% \text{ volume at the injection stage} = \frac{(PP-PT)x100\%}{(PP-PC)}$
 - $\% \text{ volume at the hold stage} = 100\% - \% \text{ volume at the injection stage}$
- 10) At what hold pressure will we get full cavities?



- between 220 and 250 bars
- 200 bars
- 300 bars

XI. Determining Cooling Stage Parameters

- **Understanding Your Product**
- **Cooling Time**
- **Mold Temperature Optimization**
- **Other Parameters That Affect Cooling**
- **How is a Cooling Experiment Optimized?**
- **Summary**

Universal Molding™ is a discipline that promotes a structure of organized events.

- Auxiliary equipment should be properly installed and correctly operating.
- Temperatures should already be reached.
- Barrel adjustments should have been programmed.
- The required closing force should have been set.
- The platen openings, their movement, speeds, and mold protection should have been carefully and precisely adjusted.
- Extended cooling time should have been set, in order to prevent it from interfering with the determination of other parameters.
- The ideal injection speed should have been determined, adjusted and should be filling about 95% of the required fill for the mold.
- The injection pressure limit should have been determined and adjusted.
- The fill balance should have been verified and adjusted.
- The hold pressure and time should have been determined and adjusted.
- The remaining cooling time should have been adjusted.

Important -- only qualified personnel who have read the operational manuals of the equipment and understand the functionality of the equipment should operate and/or make adjustments.

Understanding Your Product

Optimizing the cooling stage will depend on the type of product. Before making decisions, ask yourself the following questions:

1. What effects am I looking to optimize in the cooling stage?
2. Is shrinkage significant in the product?
3. What factors or parameters must be evaluated that could contribute to the thermal effect or effects?

There are basically two parameters that should be optimized, cooling time and cooling temperature. There are also other parameters that could affect heat removal and, consequently, the thermal dimensions and the properties of the molded product. An example of this is melt temperature;

hot melt will require more heat removal during cooling than a cooler melt would.

The melt temperature recommended by the resin manufacturer may require an adjustment. Molds that are difficult to fill may require increasing the melt temperature. Remember that changing the melt temperature means a change in viscosity. Filling cavities with thin walls could require more energy and less viscosity than cavities with thick walls, even when both are filled with the same thermoplastic.

Understand your product before deciding which parameters might be significant in the cooling stage. The thermal effect you are looking to optimize could be a function of one factor (mold temperature), two factors (mold temperature and cooling time) or three factors (mold temperature, cooling time, and melt temperature).

Cooling Time

We know that long operational cycles can be costly; consequently, the cooling time should be reduced as much as possible. To reduce the cooling time, it is recommended to carefully lower it until you get the minimum time needed to demold parts that are visually acceptable.

Extreme caution should be taken during this process; parts trapped inside the mold could result in costly damage.

In molds with cold runners, the runners could be used as a reference at the moment of demolding. The runner is usually thicker than the molded parts, and how it demolds could be used as a reference for the reduction of the cooling time.

Reducing cooling time is the first step; this does not mean that we have finished its determination. It will probably have to be modified to guarantee some dimension or property of the molded parts, even combining it with other parameters to achieve cooling optimization.

Optimization Using the Mold Temperature

This procedure is the simplest and probably where everyone should begin.

1. Determine the minimum cooling time.

Carefully, and without reaching the recovery time, reduce the cooling time until you get the least amount of time required to demold visually acceptable parts.

If the cooling time reaches the recovery time and the mold and material allow the cooling time to be further reduced, you should try to reduce the recovery time. If the process permits, reduce the recovery time by increasing recovery speed. Then, verify the melt temperature, which is likely to have increased.

Procedure for measuring melt temperature

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, retract the injection unit and purge the melt. You can do this on a removable surface for easier access.
4. Adjust the instrument to maintain the highest recorded temperature. This eliminates subjectivity when searching for the melt stabilization temperature.
5. Submerge the instrument into the melt and agitate it. When you notice the temperature starting to decrease, remove the instrument and note 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 in the same way.

Increasing the rate of recovery also increases friction, resulting in increased temperature and decreased melt viscosity. Two factors that could affect rheology are hold and fill balance. In other words, it would affect all laboratories carried out so far which is why, after increasing the rate of recovery, the temperature of the melt should be measured. If changed, you must adjust the heat zones of the barrel until reaching the temperature of the previous melt.

Again, avoid closing the mold with parts inside; this could cause expensive damage.

2. Organize and create a table for your experiment.

Select a range for your experiment, for example, $70 \pm 4^{\circ}\text{F}$. This range can be obtained from the material data sheet, from the mold manufacturer, or from a trusted molder who has molded with this material or with a similar product. Decide how many times each experiment will be repeated. Do not overload the experiment, three to five repetitions should be enough.

Include the following in the table:

- the two temperature levels.
- the number of repetitions, three for each temperature.
- spaces for the random order in which the experiments will be performed.
- spaces for the molded parts' dimensions or attributes which are being evaluated.
- spaces for the averages of the measurements.

Temp.	Rep.	Random order	Measure	Average
T_{low}	1	4	M_4	$\frac{M_4 + M_1 + M_6}{3}$
	2	1	M_1	
	3	6	M_6	
T_{high}	1	3	M_3	$\frac{M_3 + M_2 + M_5}{3}$
	2	2	M_2	
	3	5	M_5	

XI-1. Mold temperature optimization experiment table

If the attribute you are measuring is a visual and subjective effect, set a judgement scale. For example, select a person who understands the effect being measured and ask them to evaluate the effect from 1 to 4, where:

- 1 = no defect
- 2 = insignificant defect
- 3 = notable defect
- 4 = unacceptable defect

Strange as it may seem, randomizing the temperature changes in the experiments gives results that are closer to reality than when using ordered changes in temperature. Randomized changes lessen the possibility of human error.

3. Write a protocol on how to perform the laboratory.

Reduce any objectivity by writing specific procedures that everyone must follow.

Example of a simple protocol:

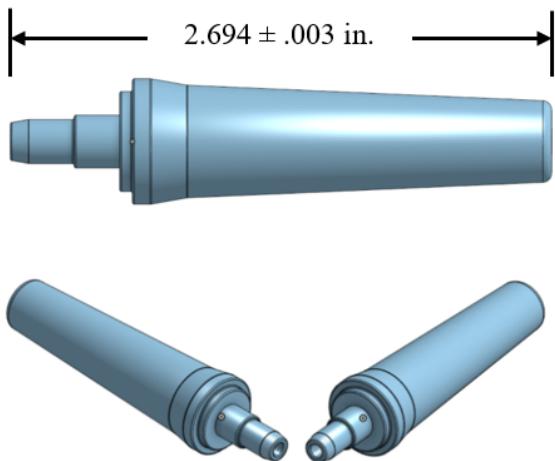
- a) Verify that the cooling water source (chiller or cooling tower) that supplies the mold's water temperature control is at least 5°C (9°F) colder than the lowest temperature you expect to program.

- b) Determine water temperature limits that correspond to the mold temperature limits. Remember that the water temperature to the mold is normally different from the temperature of the metal of the cavities.
 - c) Between experiments, stop the machine until reaching the adjusted temperature.
 - d) Once the temperature is reached and the machine is operating normally, wait 10 cycles to take the sample.
 - e) Measurements should be taken by the same person in the metrology laboratory with a calibrated instrument.
 - f) Measurements will be performed in order, for example, two hours after each demolding.
4. Perform the laboratory and enter the results in the table.
Follow the protocol procedures that were approved for this laboratory and perform your experiments, complete the table, and calculate the corresponding averages.
5. Evaluate the results.
Evaluate the results and select a temperature range that guarantees the measurement of the parts or of the effect that you are looking for.
6. Corroborate the ranges of the selected temperatures.
Inside the limits of the selected temperature, take multiple samples at distinct periods, and verify that the results comply with the prediction of the effect that is being studied.

Example:

This exercise was performed by students at the University of Puerto Rico, Mayagüez campus. The material used is low density polyethylene (LDPE) with a two-cavity mold.

The laboratory was performed in order to evaluate the effect of the mold's temperature on shrinkage. Students decided on the following critical measurement:



XI-2. Example of parts of a two-cavity mold

The resin manufacturing data sheet listed a range of 70°F to 90°F for the mold temperature.

Experiment protocol:

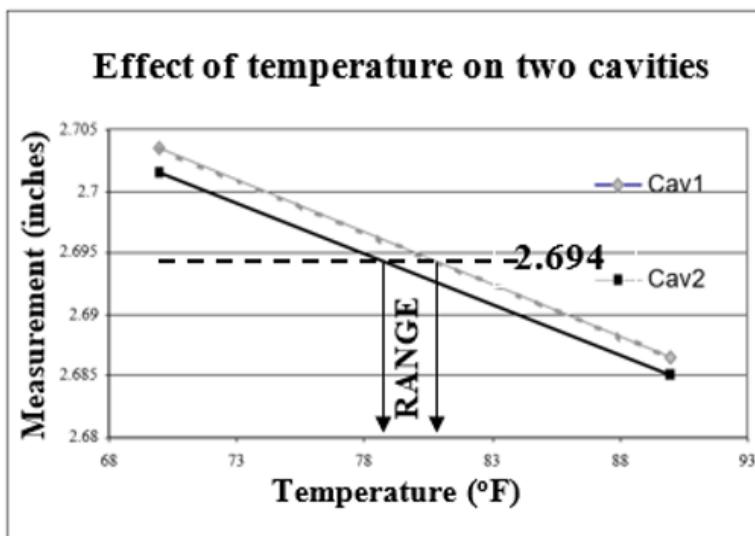
- Two temperature levels, 70°F and 90°F.
- Three repetitions per temperature level.
- Each experiment was randomized.
- Cooling time was set to 18 seconds.
- The machine was stopped between experiments until it reached the adjusted temperature.
- Once it began operating, the sample was taken after the 10th cycle.
- Measurements were taken in order, two hours after demolding.
- The same person took all the selected measurements.
- An operator verified that the mold never closed with parts trapped inside the cavities during the experiment.

Experiment results:

Mold Temperature (°F)	Average Length (inches)	
	Cavity 1	Cavity 2
70	2.7035	2.7015
90	2.6865	2.6850

XI-3. Table with effects of mold temperature on critical measurement

To understand the effect, graphs of temperature versus dimension were created, one for each cavity.



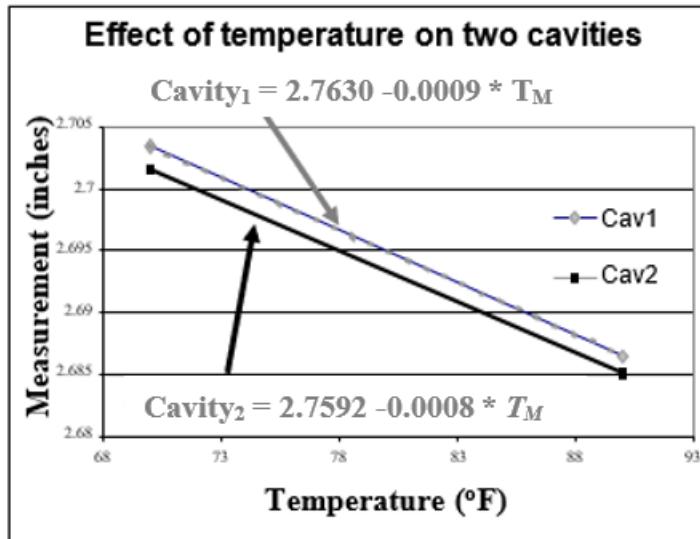
XI-4. Graph of mold temperature effect on critical measurements

The graph of length vs. temperature reveals a clear effect of the temperature on the length of the pieces. By drawing a line at the average length of 2.694 inches, we can predict that Cavity 2 should be around 78°F and Cavity 1 should be around 80°F to ensure the average length.

With a prediction equation for each cavity, we can simplify the selection of an acceptable temperature range. In this example, the equation is simple since it is the equation of a line; $Y = Y_o + MX$, where Y is the

measurement of the part, Y_0 is the intercept of the measurement coordinate, M is the slope, and X is the temperature of the mold. The value of these constants can be found using simple mathematics or with Excel.

Normally two points are enough, but if you suspect that the results are not linear, intermediate points can be used.



XI-5. Graph showing linear equations using temperatures and measurements of a two-cavity mold

By solving for temperature, we get:

$$T_{M1} = (2.7630 - \text{cavity length}_1) / 0.0009$$

$$T_{M2} = (2.7592 - \text{cavity length}_2) / 0.0008$$

Substituting the dimensional limits (2.694 ± 0.003) of 2.697 inches and 2.691 inches in both equations, we obtain the following:

	Upper dimension 2.697 in.	Lower dimension 2.691 in.
Cavity 1 temperature range	73 °F	80 °F
Cavity 2 temperature range	78 °F	85 °F
Operational Range	78 °F	80 °F

XI-6. Table with temperature limits at critical measurements of two parts

Let's evaluate the results: if the mold temperature is set to 73°F, Cavity 2 will be outside of the range, since its minimum is 78°F; if the temperature is set to 85°F, Cavity 1 will be outside of the range since its maximum is 80°F. The operational range would then be between 78°F and 80°F.

Other Parameters that Affect Cooling

Although the cooling stage's control parameters are cooling time and mold temperature, we must recognize that the melt temperature contributes as well.

Imagine that, after performing the previous mold temperature laboratory, a product requires better thermal dimensional control. A decision was made to optimize these thermal dimensions by evaluating the contribution of three factors:

T_M = temperature of the mold

t = cooling time

T_m = temperature of the melt

The critical dimensions could be represented by the following linear equation:

Critical dimension, D_C =

$$\bar{D}_C + \beta_0 T_M + \beta_1 t + \beta_2 T_m + \beta_3 T_M t + \beta_4 T_M T_m + \beta_5 t T_m + \beta_6 T_M T_m t$$

where:

\bar{D}_C = average measurement

$\beta_0 T_M$ = effect of the mold temperature

$\beta_1 t$ = effect of the cooling time

$\beta_2 T_m$ = effect of melt temperature
 $\beta_3 T_m t \dots \beta_6 T_M T_m t$ = combined effects
and the symbols β_o to β_6 are constants.

This type of exercise assumes that the behavior is linear. If the **Universal Molding™** laboratory is performed in the recommended order, those who have a good understanding of the morphology of thermoplastics, thermal dimensions, and mass dimensions will be able to select reasonable experimental ranges for each factor.

If the experimentation ranges are very large, e.g. $\pm 10^{\circ}\text{F}$ for the melt temperature, you can be sure that the results would be nonlinear, and a more complex equation would be required.

The selection of factors is important; assigning an effect to multiple factors, such as temperature of the mold, cooling time, temperature of the melt, and hold pressure, without fundamentals would complicate and make the laboratory more expensive. Most effects are corrected by experimenting with a single parameter. For example, a critical dimension in a semi-crystalline material is likely to be controlled with the temperature of the mold. Avoid complicating the cooling laboratory.

Let's look at another example where we suspect that the hold pressure has a significant effect on the dimensions of the product. In this laboratory, it is assumed that the critical dimensions, in addition to being a thermal function, are also a function of the quantity of material (mass dimensions) and should consider two factors:

T_M - temperature of the mold
 P - hold pressure

$$\text{Critical dimension, } D_C = \bar{D}_C + \beta_0 T_M + \beta_1 P + \beta_2 T_M P$$

Where:

\bar{D}_C = average measurement
 $\beta_0 T_M$ = effect of mold temperature
 $\beta_1 P$ = effect of hold pressure
 $\beta_2 T_M P$ = combined effect

A combined effect is when two or more factors, or parameters, join to influence the effect being evaluated. A hypothetical example: two employees work separately, doing the same job. Their job is packing molded products into boxes, and each employee packs 50 boxes a day. One day they decide to work as a team; one assembles the boxes and fills them with product, the other seals them closed and stacks them. At the end of the day, they pack 120 boxes. The combined effect was to pack 20 more boxes per day. This also happens when two parameters combine and create an effect on the molded product.

How is a Cooling Experiment Organized?

1. Establish an objective and the effect, or effects, to be measured.
2. Identify which factors or parameters are presumed to have some effect and how they are related to the effect to be examined. Selecting these factors is probably the most delicate part of this process. Include only those factors that have a high probability of influencing the effect being evaluated on the molded product. An exercise of more than one parameter or factor consumes time and resources; be careful with your design or you could end up with inconclusive results. Most cooling laboratories are carried out with a single factor, mold temperature.
3. Select the limits, or ranges, of each factor. Here you must be quite judicious; too large a range could create nonlinear results, create inconclusive results, or could complicate the experiment. Too small a range could reveal no contribution to the effect, even though some may exist.
4. Create a table for your results; include spaces for the variables of the effects and their combinations, the number of repetitions, and number of experiments.

Process Factors		Repetitions		
		I	II	III
1	20	4	1	2
2	20	12	4	5
3	25	4	7	8
4	25	12	10	11
				12

XI-7. Table of cooling parameters (temperature and time), with their combinations and their repetitions

5. Write a protocol on how to perform the lab.
6. Perform the laboratory, and take samples as indicated in the written protocol.
7. Evaluate the data and obtain results. If the laboratory uses two factors (ex: cooling time and mold temperature) with a mold that contains only a few cavities (four or less), data analysis could be performed with Excel. However, when there are more than two variables and more than four cavities, you will need to use a statistical program.

Results should reveal significant and insignificant factors, as well as significant and insignificant interactions.

Summary

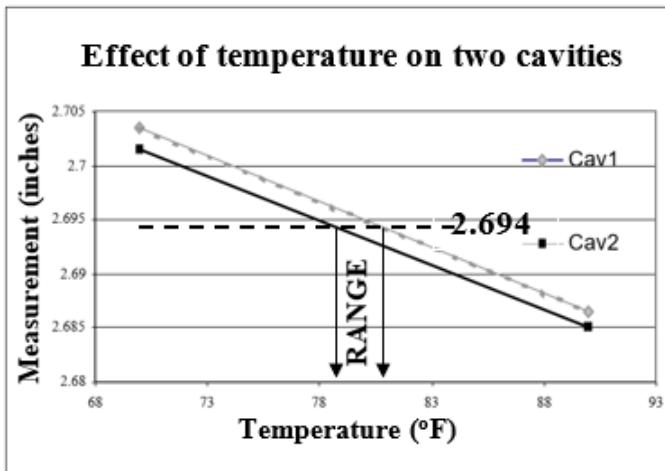
- Begin the optimization of the cooling stage by finding the minimum cooling time that produces parts that can be demolded and are visually acceptable.
- The first laboratory should only be performed with the mold temperature.
- Consider a second factor only when the cooling stage cannot be optimized with the mold temperature.
- When the effect is an attribute, for example, a visual or physical defect, select one person who knows how to evaluate the effect, and establish some numerical evaluation system that represents the criteria for acceptance.

- In order to reduce the possibility of objectivity on the part of those performing the laboratory, create a written, step-by-step protocol that must be followed.
- If the laboratory is for two factors (e.g. cooling time and mold temperature) with a mold that only has a few cavities (four or less), data analysis can be created in Excel; with more than two factors and more than four cavities, use a statistical program.

Questions

- 1) When optimizing the cooling stage,
 - a. the cooling time is set so it is equal to the hold time.
 - b. the pressure and hold time have been determined and adjusted.
 - c. the melt temperature remains fixed since it does not affect heat removal or thermal dimensions.
- 2) When organizing a mold temperature optimization experiment, include more than ten repetitions per experiment in order to obtain more accurate results.
 - a. True, the experiment improves when the number of repetitions is increased.
 - b. False, do not overload the experiment; three to five repetitions should be enough.
- 3) Select all the correct statements. While optimizing the cooling stage,
 - a. long operational cycles could be costly; consequently, cooling time should be minimized while still guaranteeing demolding and part quality.
 - b. extreme caution should be exercised because parts trapped inside the mold can cause costly damage.
 - c. in a conventional mold, you must ensure that the cooling time is always greater than the recovery time.
 - d. All the above.
- 4) In a cooling optimization experiment, parameter adjustments made during data compilation should be randomized.
 - a. True, since random changes lessen errors as a result of human convenience.
 - b. False, since random changes complicate the laboratory and do not contribute to the experiment.
- 5) Select all the correct statements. In a cooling optimization experiment,
 - a. the chiller or cooling tower that supplies water to the temperature control of the mold must be at least 5°C (9°F) cooler than the lowest temperature you expect to program.
 - b. the temperature of the water for the mold is normally equal to the temperature of the metal in the cavities.

- c. you should verify the results with values outside of the operational ranges obtained from the experiment.
- 6) The graph demonstrates the effect of the mold temperature on one dimension of the molded parts. For a dimension of 2.694 inches, what would be the recommended temperature?



- a. 80°F
 b. 83°F
 c. 73°F
- 7) If you decide to optimize cooling dimensions by evaluating the contribution of three factors,
- a laboratory of three parameters or factors consumes time and resources; be careful in your design or you could end up with inconclusive results.
 - when one of the factors is hold pressure, you should assume that the evaluated dimension is a function of mass dimensions.
 - when one of the factors is melt temperature, you should assume that the evaluated dimension is a function of mass dimensions.
 - both a and b are correct.
- 8) In a two-factor experiment for mold temperature (T_M) and melt temperature (T_m),

$$\text{Critical dimension, } D_C =$$

$$\bar{D}_c + \beta_0 T_M + \beta_1 T_m + \beta_3 T_M T_m$$

It was found that mold temperature (T_M) and melt temperature (T_m) were significant and that their combined effect was not significant. The equation could then be reduced to

a. $\bar{D}_c = \bar{D}_c + \beta_1 T_m + \beta_3 T_M T_m$

b. $\bar{D}_c = \bar{D}_c + \beta_0 T_M + \beta_1 T_m$

c. $\bar{D}_c = \beta_0 T_M + \beta_1 T_m$

- 9) In an experiment with three factors, mold temperature (T_M), melt temperature (T_m), and cooling time (t), the following linear equation is used:

Critical dimension, $D_C =$

$$\frac{\bar{D}_c + \beta_0 T_M + \beta_1 t + \beta_2 T_m + \beta_3 T_M t + \beta_4 T_M T_m + \beta_5 t T_m + \beta_6 T_M T_m t}{() () () () () ()}$$

Write the number of each description below its corresponding component of the equation.

1. average measurement
2. effect of cooling time
3. combined effects
4. effect of melt temperature
5. effect of mold temperature

XII. Process Limits

- **Maximum Injection Pressure Limit**
- **Cushion Lower Limit**
- **Cushion Upper Limit**
- **Recovery Time Limit**
- **Back Pressure 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. There are many consequences; use these limits appropriately, and you will enjoy their benefits and savings.

What would cause the maximum injection pressure limit to be reached, and what could be its consequences?

Reaching the maximum injection pressure limit could be a result of a blocked cavity, either because a part was trapped in the cavity or there was a plugged gate. 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. Another possibility could be that the recovery position was increased, causing excessive fill during the injection stage, significantly higher than the 95% of the mold's capacity.

What would cause the cushion lower 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, to reach the wiring of the hot runner system, to leak between mold actuators and damage the mold; in general, it could cause substantial damages. In addition, if the cushion position reaches zero (totally empty injection unit) it will nullify the hold stage, and part dimension changes could be observed.

What would cause the cushion upper 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. Also, if it's a hot-runner mold, it could be that a hot tip is clogged or damaged. 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.

Inconsistent resin flow may be due to the barrel heat affecting the feed throat, causing material agglomeration. This problem can arise for several reasons, such as an excessively high barrel temperature or poor cooling of the feed throat.

What is an appropriate upper and lower limit for back pressure?

Before providing an opinion, let's start with an overview. During recovery, 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 recovery 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, 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.

Questions

1. A process fills 97% in the injection stage and creates flash. The procedure to verify and resolve the situation is:
 - a) After turning off the holding stage, the flash continues, identifying that too much material is being filled in the injection stage. Gradually reduce the recovery position until the flash defect disappears. To prevent the situation from recurring, set a high recovery limit.
 - b) After turning off the holding stage, the flash continues, indicating that too much material is being filled in the injection stage. Gradually reduce the transfer position until the defect disappears.
2. In a process, the maximum injection pressure alarm was triggered. The reason could have been
 - a) a blocked cavity or the injection stage was set to fill too little, less than 95%.
 - b) a blocked cavity or the injection stage was set to fill too much, more than 95%.
3. What would cause the cushion lower limit to be reached?
 - a) Dirty or defective check ring, melt leaking between the sprue bushing and the nozzle tip, or the melt is leaking into the mold, either as flash leakage between mold actuators or within the hot runner system.
 - b) The recovery position was set too high, causing over 95% filling in the injection stage, or the transfer position was set too high, causing over 5% filling in the holding stage.
4. What would cause the cushion upper limit to be reached?
 - a) Blocked cavity or an obstructed runner, and if it is a hot runner mold, a hot tip could be obstructed or damaged.
 - b) Dirty or defective check ring, melt leaking between the sprue bushing and the nozzle tip, or the melt is leaking into the mold.
5. What would cause the recovery time limit to be reached?
 - a) Melt leaking between the sprue bushing and the nozzle tip.

- b) Inconsistent resin flow due to the feed hopper running out of resin.
Additionally, it could be because the barrel heat is affecting the feed throat, causing material agglomeration.

Appendices

I – Troubleshooting

II – Universal Mold Data

III – *Universal MoldingTM* Equations

IV – General Procedures for *Universal MoldingTM*

V – English Terms in Spanish

VI – Spanish Terms in English

VII – Operational Costs

I - Troubleshooting

During troubleshooting, experience plays an important factor. In the event there is a problem that you cannot solve, seek help. Do not sacrifice production or quality; ask. This list is only a reference; use it judiciously.

Brittle parts - The parts become brittle or break

Possible Causes	Actions
1. Resin is too cold	1.a. Increase backpressure. 1.b. Increase melt temperature.
2. Material degradation in the barrel	2.a. Reduce melt temperature. 2.b. Reduce backpressure. 2.c. Reduce the injection rate. 2.d. Purge, if necessary.
3. Material contamination	3.a. Verify material in the hopper. 3.b. Purge, if necessary.
4. Material degradation during the drying process	4. Decrease dryer time and/or temperature.
5. Moisture in the material	5. Verify moisture content, dry properly.

Bubbles (voids) - Air trapped inside the part

Possible Causes	Actions
1. Moisture in the material	1. Check moisture content, dry properly.
2. Material is too hot	2. Decrease the melt temperature by adjusting to a suitable barrel temperature profile.
3. Inadequate venting	3. Ensure that the mold has adequate and clean vents.
4. Internal bubbles caused by shrinkage	4.a. Increase backpressure and/or hold pressure. 4.b. Decrease the melt temperature.

Weld line - Lines on the part formed by two or more melt flows joining together

<i>Possible Causes</i>	<i>Actions</i>
1. Low mold temperature	1. Increase mold temperature.
2. Material is too cold	2. Increase melt temperature.
3. Low injection rate	3. Increase speed. Injection time should be significantly reduced.
4. Humid resin	4. Dry material properly.

Fading - Inadequate color

<i>Possible Causes</i>	<i>Actions</i>
1. Degraded material in the barrel	1. Purge the barrel.
2. High melt temperature	2. Decrease melt temperature by adjusting to a suitable barrel temperature profile.
3. Contaminated material	3. Check the material.
4. Inadequate vents	4. Clean existing vents or ventilate mold properly.

Burns - Marks on the part due to degradation

<i>Possible Causes</i>	<i>Actions</i>
1. High injection speed	1. Decrease injection speed.
2. High backpressure	2. Decrease backpressure.
3. Inadequate vents	3.a. Verify that there are vents. 3.b. Clean vents.
4. Problems in mold design (material suffers friction, causing degradation)	4.a. Change the location of the gate. 4.b. Ensure that the part has generous radii (no sharp corners).
5. Nozzle hole is too small or clogged	5. Replace or clean the nozzle.
6. High screw recovery	6. Decrease screw recovery speed.
7. High melt temperature	7. Decrease the melt temperature by adjusting to a suitable barrel temperature profile.

Cloudiness - Cloudy appearance in parts (more noticeable in clear parts)

<i>Possible Causes</i>	<i>Actions</i>
1. Material contamination	1.a. Check material and change if necessary. 1.b. Increase melt temperature.
2. Gases or moisture in the resin	2.a. Dry material properly. 2.b. Ventilate mold properly.
3. Material is too cold	3. Increase the melt temperature.
4. Mold is too cold	4. Increase mold temperatures.
5. Mold release	5. Eliminate the use of mold release.

Flash - Excess plastic around the parting line

<i>Possible Causes</i>	<i>Actions</i>
1. High hold pressure	1. Decrease hold pressure.
2. Mold is too hot	2. Lower mold temperatures.
3. Inadequate closing force	3. Increase tonnage.
4. High melt temperature	4. Lower the melt temperature.
5. Late transfer position to hold	5. Adjust to an appropriate transfer position and compensate by adjusting the same distance to the recovery position.
6. Material with moisture	6. Improve drying.

Flow lines - Marks in the direction of the melt flow

<i>Possible Causes</i>	<i>Actions</i>
1. Low mold temperature	1. Increase the mold temperature.
2. Material is too cold	2. Increase the melt temperature.
3. Inadequate runner/gate	3. Check the size of runner and gates, and request a redesign.
4. High injection speed	4. Decrease injection rate.
5. Humid resin	5. Dry material properly.

Worm-shaped jetting on the surface of the part

<i>Possible Causes</i>	<i>Actions</i>
1. Gates too small	1. Verify and request a gate redesign.
2. Poorly localized gate	2. Request a redesign.
3. Injection speed too high	3. Slow down the injection rate.
4. Small nozzle hole	4. Replace nozzle.

Surface delamination – Layer separation on the surface of the pieces

Possible Causes	Actions
1. Contaminated material	1. Check the material and replace, if necessary.
2. Low melt temperature	2. Adjust to a suitable barrel temperature profile.
3. Melt mixture is not uniform	3. Increase backpressure.
4. Low mold temperature	4. Increase mold temperature.
5. Low injection rate	5. Increase speed. Injection time should be significantly reduced.

Incomplete Shot - Parts are not completely filled

Possible Causes	Actions
1. Low hold pressure	1. Increase hold pressure.
2. Short hold time	2. Increase hold time.
3. Unbalanced cavities	3. Balance the fill and redesign the mold, if necessary.
4. Low melt temperature	4. Increase the melt temperature by adjusting to an appropriate temperature profile.
5. Small runners/gates	5. Request a redesign of runners and/or gates.
6. Low mold temperature	6. Increase mold temperature.
7. Low recovery position	7. Verify that the transfer position is adequate, and then increase the recovery position.

Sink - Depressions or sunken areas in sections of the parts

Possible Causes	Actions
1. Low injection rate	1. Increase speed. Injection time should be significantly reduced.
2. Inadequate design of the mold and/or part	2. Redesign part (uniform wall thicknesses are required).
3. Low melt temperature	3. Increase melt temperature with a suitable temperature profile.
4. Gas trapped in the mold	4. Vent the mold properly.
5. Low hold pressure	5. Increase hold pressure.
6. Excessive mold temperature causes shrinkage	6. Lower the temperature of the mold.
7. Low hold time	7. Increase hold time.
8. Low mold temperature causing premature gate freezes	8. Increase the temperature of the mold.

The parts stick to the cavity

Possible Causes	Actions
1. Scratched cavity	1. Polish in the direction of demolding.
2. Static	2. Demagnetize the cavity.
3. High hold pressure	3. Decrease the hold pressure.
4. Short cooling time	4. Increase cooling time.
5. Shrinking in the wrong direction	5. Set the temperature of the core so it is higher than the cavity temperature.
6. Insufficient draft and/or detachment angle	6. Consider changes in part and/or mold design.

The parts stick to the core

Possible Causes	Actions
1. High hold pressure	1. Decrease the hold pressure.
2. High core temperature	2. Adjust the mold temperatures.
3. High melt temperature	3. Lower the melt temperature with a suitable temperature profile.
4. Insufficient draft and/or detachment angle	4. Consider changes in part and/or mold design.
5. Static	5. Demagnetize the cavity.

Silver lines - Imperfections on the surface of the parts

Possible Causes	Actions
1. Moisture in the resin	1. Dry the material properly.
2. Obstruction in the nozzle	2. Clean the nozzle.
3. High injection speed	3. Slow down the injection rate.
4. High melt temperature	4. Lower the melt temperature with an appropriate barrel temperature profile.
5. Nozzle is too hot	5. Lower nozzle temperature.
6. Contaminated resin	6. Check the material; replace it if necessary.
7. Gates are too small	7. Increase gate size.

Buckling - Twisting or curving of the parts due to uneven shrinkage

<i>Possible Causes</i>	<i>Actions</i>
1. Hot parts upon ejection	1.a. Lower the mold temperature. 1.b. Increase the cooling time.
2. Uneven part cooling	2. Adjust the temperatures of the mold faces.
3. Non-uniform wall thickness	3. Redesign the part.
4. Parts are overpacked	4. Decrease the hold pressure.

Maximum injection pressure limit alarm

<i>Possible Causes</i>	<i>Actions</i>
1. Blocked cavity	1. Stop the machine. Check and clean the mold.
2. Injection stage set to fill too much.	2. Reduce the recovery position to about 95% of the mold.

Cushion lower limit alarm

<i>Possible Causes</i>	<i>Actions</i>
1. Dirty or defective check ring	1. Check and clean or replace.
2. Melt leaking between sprue bushing and nozzle tip	2. Check if any of the components are scratched and/or crushed, if the hole diameters are incorrect, or if the contact radii are incorrect. If so, replace them.
3. The melt is leaking into the mold, either as flash, filtration between actuators, or within the hot runner system.	3. Check, and if so, make the repair or correction as soon as possible. Failure to do so could result in extensive damage to the mold.

Cushion upper limit alarm

<i>Possible Causes</i>	<i>Actions</i>
1. Blocked cavity or a clogged gate.	1. Check and clean.
2. If it's a hot-runner mold, it could be that a hot tip is clogged or damaged.	2. Check, clean or replace.

Recovery time limit alarm

<i>Possible Causes</i>	<i>Actions</i>
1. Inconsistent resin flow, which may be due to the heat from the barrel affecting the feed throat, causing material to clump.	1a. Reduce the temperature of the heat zone near the feed throat. 1b. Check the water flow in the throat of the barrel. If it's clogged, correct it. 1c. Check the temperature of the water in the throat of the barrel and reduce it if it is too high.
2. The feed hopper is running out of resin.	2. Check and correct the material supply.

II - Universal Mold Data

Remember that these parameters are for the mold and the plastic. To be used they must be transferred or converted into injection machine parameters.

1. mold name and number
2. name and type of material
3. filling time to get a fill of about 95%
4. weight of the parts at the time of transfer, with the hold turned off
5. plastic pressure at the time of transfer
6. total cycle time
7. hold time
8. hold pressure
9. total weight or total injection volume
10. cooling time
11. mold temperatures
12. flow and water to the mold
13. water temperatures entering the mold
14. water temperatures exiting the mold
15. water pressures entering the mold
16. water pressures exiting the mold
17. melt temperature entering the mold
18. time it takes to open the mold and eject the parts
19. time to completely close the mold
20. recovery volume

III - Universal MoldingTM Equations

1- Drying

Drying hopper volume:

To determine the size of a dryer hopper, you need to know:

- the material consumption in lb/hr or kg/hr
- the drying time in hours
- the bulk density of the plastic resin in lb/ft³ or kg/m³.

I. With virgin material:

$$\text{Volume} = \text{consumption} \times \frac{\text{drying time}}{\text{bulk density}}$$

II. With virgin and regrind material:

$$\text{Volume} = T * C * \left(\frac{\%V}{D_{\text{virgin}}} + \frac{\%R}{D_{\text{regrind}}} \right)$$

Where:

T = drying time (hours)

C = resin consumption (lb/hr or kg/hr)

D_{virgin} = virgin material density (lb/ft³ or kg/m³)

D_{regrind} = regrind density (lb/ft³ or kg/m³)

$\%V$ = % of virgin

$\%R$ = % of regrind

Dry air flow from the dryer:

Flow = material flow factor * consumption

$$\begin{aligned} &\text{Maximum material factor} \\ &= 1 \text{ cfm} / (1 \text{ lb of material/hr}) \\ &= (63 \text{ liters/min}) / (1 \text{ kg of material/hr}) \end{aligned}$$

Delta temperature:

$$\begin{aligned} \Delta T_{\text{Fahrenheit}} &= 9/5 * \Delta T_{\text{Celsius}} \\ \Delta T_{\text{Celsius}} &= 5/9 * \Delta T_{\text{Fahrenheit}} \end{aligned}$$

Recovery consumption:

$$\text{Recovery consumption} = \\ (\text{injection shot weight}) / (\text{recovery time})$$

Material consumption:

$$\text{Material consumption} = \\ (\text{total amount of material}) / (\text{process cycle time})$$

2- Cooling

Removed heat:

$$\text{Removed heat} = \\ \frac{(\text{total quantity of material}) * (\text{energy required for the material})}{(\text{process cycle time})}$$

Empirical equation to determine the required water flow of a chiller:

$$gmp = \frac{24 * (\text{chiller tons})}{\Delta T}$$

Where:

- gpm = gallons of water per minute
- chiller tons = chiller tons required to cool the mold
- Delta T (°F) = outlet water temperature – inlet water temperature

Extended cooling time:

$$\text{Extended cooling time} = \\ \text{recommended cooling time} \times 1.3$$

Calculated method to determine the cooling time (E):

$$E = -\frac{G^2}{2\pi\alpha} \ln \ln \left(\frac{\pi}{4} \frac{(T_x - T_M)}{(T_m - T_M)} \right)$$

Where:

T_x = deflection temperature

T_M = mold temperature

T_m = melt temperature

G = part thickness
 α = thermal diffusivity

Material	α	T_m (°F)	T_M (°F)	T_x (°F)
ABS	0.000185	475	135	203
CA, CAP	0.000181	400	110	192
CAB	0.0002	400	110	201
HIPS	0.000059	440	85	185
IONOM	0.000148	440	85	125
LDPE	0.000176	390	75	113
MDPE	0.000194	340	75	155
HDPE	0.000217	480	75	186
PA 6, 6/6	0.000109	530	150	356
PC	0.000132	560	180	280
PET	0.000138	540	120	153
PP	0.000077	470	105	204
PPO/PS	0.000144	530	185	234
PPS	0.000166	630	210	210
PS g.p.	0.000087	420	85	180
PSU	0.000149	700	250	345
PVC	0.000107	380	85	156
PVCrig	0.000123	380	85	174
SAN	0.000088	450	150	225

Pressure loss (Delta P) and water temperature loss (Delta T):

$$\text{Delta } P = \text{inlet water pressure} - \text{outlet water pressure}$$

$$\text{Delta } T = \text{outlet water temperature} - \text{inlet water temperature}$$

3- Clamp

Clamping force:

Clamping force = (pressure factor of the melt) x (projected area)

US ton = 2000 lbf

Metric ton = 1.10 x US ton

US ton = 8.90 Kilo-Newtons (kN)

Metric ton = 9.81 Kilo-Newtons (kN)

Material	US ton/in ²		kN/cm ²	
Polypropylene	1.5	3.5	2.1	4.8
High density polyethylene	1.5	2.5	2.1	3.5
Low density polyethylene	1.0	2.0	1.4	2.8
Nylon 66	3.0	5.0	4.1	6.9
Polycarbonate	3.0	5.0	4.1	6.9
Flexible PVC	1.5	2.5	2.1	3.5
Rigid PVC	2.0	3.0	2.8	4.1
Polystyrene	2.0	4.0	2.8	5.5

Thin wall calculation, TW:

A thin wall calculation is a factor that represents fill difficulty. This factor takes into consideration the distance that the melt must travel and how narrow those passages are. This factor is represented by the following equation:

$$\text{Thin wall calculation} = \frac{(\text{farthest flow path})}{(\text{thinnest wall on the path})}$$

TW	Criteria
≥ 200	Use the highest pressure factor. Force = (projected area) x (highest pressure factor)
≤ 100	Use the smallest pressure factor. Force = (projected area) x (smallest pressure factor)
between 100 and 200	Interpolate between pressure factors. Force = (projected area) x (interpolated pressure factor)

Lateral action resulting in additional clamping force:

These wedges are manufactured at an angle, φ , and only a fraction of this lateral force will be seen in the direction of the closing wedges. This resulting force in the direction of the press is determined by multiplying the lateral force by the tangent of the wedge's angle:

$$\text{Force as a result of lateral action} = \text{lateral force} \times \tan(\varphi)$$

4- Injection unit

Density and specific density:

$$\text{Density} = \frac{\text{mass}}{\text{volume}}$$

$$\text{Specific density} = \frac{\text{material density}}{\text{water density}}$$

Where the density of water is 1 g/cm³ at room temperature.

Injection speed and flow:

$$\text{Injection speed} = \frac{\text{distance}}{\text{time}}$$

$$\text{Injection flow} = \frac{\text{volume}}{\text{time}}$$

$$\text{Injection flow} = \frac{(\text{screw diameter})^2 * \frac{\pi}{4} * (\text{displacement})}{\text{time}}$$

Barrel utilization:

Barrel utilization (%U) is a comparison between the maximum capacity of the injection unit and the capacity required to fill the mold.

$$\%U = \% \text{ of Utilization} = \frac{(\text{volume used})}{(\text{volume the barrel is capable of})} * 100\%$$

Where:

volume used = the volume programmed to the mold's requirements
volume the barrel is capable of = the maximum recovery volume of the screw

Recovery position:

The recovery position is the place the screw should reach in order to fill the mold.

$$\begin{aligned} \text{Recovery position} = \\ \text{transfer position} + \text{injection displacement} \end{aligned}$$

Combining the equations of weight, density, and volume, we summarize:

$$\begin{aligned} \text{Injection displacement} &= \frac{1.27W}{D^2} \\ \text{Recovery position} &= \text{transfer position} + \frac{1.27W}{D^2} \end{aligned}$$

Where:

ρ = specific density of the melt (gr/cm³)

W = weight of parts and runner (gr)

D = diameter of the injection screw (cm)

Note that this equation does not consider that, during injection, only about 95% of the mold is filled. This excess is ignored due to the fact that, during the injection stage, some material always sneaks to the other side of the check ring, whether during check ring closure or as a result of leaks between the check ring and the barrel.

Discharge density:

This density is more precise for determining the recovery positions, since it considers several factors:

- mass
- volume
- melt temperature
- back pressure
- melt leaks through the check ring during injection.

Discharge density is calculated in an existing process by measuring the injected volume and the total injected weight. The injected volume is determined using the cylinder equation:

$$\text{Volume} = \text{area} \times \text{length}$$

Where:

$$\text{Area} = (\text{screw diameter})^2 * \frac{\pi}{4}$$

$$\text{Length} = \text{recovery position} - \text{cushion position}$$

The injection weight is determined by weighing the molded parts plus the runner (if one exists).

$$\text{Discharge density} =$$

$$\frac{\text{total injection weight}}{((\text{screw diameter})^2 \times \pi/4) \times (\text{recovery position} - \text{cushion position})}$$

If the discharge density is known, the injection displacement and the recovery position equations would look like this:

$$\text{Injection displacement}_{95\%} \text{ XE "desplazamiento de inyección"} = 95\% \frac{1.27}{\rho_d L}$$

$$\begin{aligned} \text{Recovery position} = \\ \text{transfer position} + 95\% * \frac{1.27W}{D^2} \end{aligned}$$

Where:

$$\rho_d = \text{discharge density (g/cm}^3)$$

$$W = \text{weight of the parts with the runner (g)}$$

$$D = \text{diameter of the injection screw (cm)}$$

Rheology by power:

Peak power – The maximum power reached by the injection unit, usually at the transfer position (change from injection to hold).

$$\begin{aligned} \text{Peak power} = \\ \text{average injection flow} \times \text{pressure at the transfer position} \end{aligned}$$

Average injection flow – This flow is a function of the volume injected during the injection stage and the injection time.

Average injection flow =
injection volume / injection time

Rheology by viscosity:

Change in speed V_x , in the direction of Y ;

$$\text{Shear rate} = \dot{\delta} = \frac{\text{change in speed}}{\text{distance}} = \frac{\Delta V_x}{\Delta Y}$$

Shear stress = viscosity x shear rate

$$\tau = \mu \times \dot{\delta}$$

Relative viscosity = plastic pressure x injection time

$$\text{Relative shear rate} = \frac{1}{\text{injection time}}$$

Intensification ratio, R_i =
plastic pressure, P_p / hydraulic pressure, P_H

$$\begin{aligned}\mu_R &= P_H R_i T \\ \mu_R &= P_P T\end{aligned}$$

Approximated rheology:

Initiation of the injection time plateau

$$T_{plateau} = T_{min} + (T_{max} - T_{min}) / 9$$

Conventional molding industries

$$T_{intermediate} = T_{min} + (T_{max} - T_{min}) / 18$$

Industries that mold sensitive materials

$$T_{sensitive} = T_{min} + (T_{max} - T_{min}) / 12$$

High-volume injection industries

$$T_{fast} = T_{min} + (T_{max} - T_{min}) / 36$$

Where:

T_{min} - injection time for the maximum injection speed

T_{max} - injection time for the minimum injection speed

$T_{plateau}$ - injection time where the plateau starts on the graph

Fill balance:

Sum of the weight of all cavities:

$$W_T = \sum_{\# cav.}^1 W_i, i = 1 \text{ to } \# \text{ of cavities}$$

Volume deviation for each cavity, Vd_i

$$Vd_i = \left[\frac{\frac{W_i}{W_T}}{\left(\frac{W_T}{\# cavities} \right)} - 1 \right] 100\%, i = 1 \text{ to } \# \text{ of cavities}$$

% injection volume by stage:

$$\begin{aligned} \text{ % volume at injection stage} &= \\ \frac{\text{recovery position} - \text{transfer position}}{\text{recovery position} - \text{cushion position}} * 100 \end{aligned}$$

If you are using a control system with **Universal** parameters and working with volume instead of position, apply this formula:

$$\begin{aligned} \text{ % volume at injection stage} &= \\ \frac{\text{recovery volume} - \text{transfer volume}}{\text{recovery volume} - \text{cushion volume}} * 100\% \end{aligned}$$

$$\begin{aligned} \text{ % volume at hold stage} &= \\ 100\% - \text{ % volume at injection stage} \end{aligned}$$

IV - General Procedures for *Universal Molding*TM

I. Sizing and Initial Data

1. Determine the clamping force _____
2. Determine the required injection volume _____
3. Select an approximated total cycle _____
4. Determine the approximate resin consumption per hour _____

5. Resin brand and type _____
6. Colorant brand and type _____
7. Colorant % _____
8. Regrind % _____

II. Auxiliary Equipment

1. Water temperature control unit
Determine the water flow to the mold _____
Select an initial water temperature _____
2. Dryer
Size the hopper dryer volume _____
Size the dry air flow _____
Drying temperature _____
3. Colorant additive
% colorant required _____
Determine the consumption of colorant/hour _____
4. Regrind additive
% regrind required _____
Determine the consumption of regrind/hour _____

III. Mold and Machine Sizing

1. Horizontal distance _____ < _____ between bars
2. Vertical distance _____ < _____ between bars
3. Closed distance _____ > _____ minimum opening
4. Open distance _____ < _____ maximum opening
5. Ejector pattern _____ = _____

IV. Injection Molding Machine Sizing

1. Injection unit
Determine % of utilization, %U _____
Determine the transfer position _____

Determine the corresponding temperature profile _____ / _____ / _____

Determine the backpressure. Ex: 750 psi plastic (machine = plastic/ R_p)

Determine the approximated relative position _____

2. Nozzle tip

Length _____

Hole diameter _____

Contact radius _____

3. Sprue bushing

Hole diameter _____

Contact radius _____

V. Initial process setup

1. Start and set the auxiliary equipment parameters

Dryer

Water temperature controller

Additive feeder

Hot runner temperature control

2. Injection unit

Start and set the barrel temperature zones

Set the backpressure

Set the recovery speed

Set the approximated recovery position

Set the extended cooling time

3. Press setup

Set mold opening positions and speeds

Set the mold protection

Set the movements of the ejectors

Set the movements of the cores, if applicable

VI. Determination of Machine Parameters

(once the auxiliary equipment is ready and temperatures have been reached)

1. Fill

Determine the injection pressure limit _____

Determine the ideal injection time _____

Readjust the injection volume to about 95%

Note the final recovery position _____

Perform the flow balance lab

2. Hold
Determine the hold pressure _____
Determine the hold time _____
3. Cooling
Determine the water temperature of the mold _____
Fixed/Moving _____ / _____
Determine the cooling time _____
4. Recovery
Adjust the recovery speed according to the new cooling time
Note the recovery time _____
5. Recalculate the auxiliary equipment according to the new cycle time

VII. Convert to Universal Parameters

Auxiliary Equipment

1. Water temperature control (gpm/lpm) to mold _____
Water temperatures to mold Fixed/Movable _____ / _____
2. Dryer
Hopper volume _____
Dry air flow _____
Drying temperature _____
3. Additive feeder, % of colorant _____
Color consumption per hour _____
4. Regrind ratio, % _____
Regrind consumption per hour _____

Mold Data

1. Horizontal distance _____
2. Vertical distance _____
3. Closed mold height _____
4. Open mold height _____
5. Ejector pattern _____
6. Material _____
7. Color additive _____

Injection Molding Machine – (m) machine / (u) Universal

1. Clamping force _____
2. Total cycle time _____
3. Material consumption per hour _____

4. Press platens

Horizontal distance between tie bars _____

Vertical distance between tie bars _____

Verify the ejectors pattern _____

Maximum opening _____

Minimum opening _____

5. Press setup

Mold opening position _____

Mold opening and closing time _____

6. Injection

% of barrel utilization _____

Injection pressure limit (m) _____ (u) _____

Ideal injection velocity (m) _____ (u) _____

Transfer position (m) _____ (u) _____

Recovery position (m) _____ (u) _____

Temperature profile _____ / _____ / _____ / _____

7. Hold

Hold pressure (m) _____ (u) _____

Hold time _____

8. Cooling

Water temperature of the mold Fixed/Movable _____ /

Cooling time _____

9. Recovery

Recovery speed _____

Recovery time _____

Backpressure (m) _____ (u) _____

V – English Terms in Spanish

auger	tornillo sin fin
backpressure	contrapresión
barrel	barril
barrier screw	tornillo con barrera
blower	bomba
boost to hold	de inyección a empaque
cavity	cavidad
check ring	anilla
chiller	equipo de refrigeración
cold slugs	pedazos fríos
cores	noyos
cushion	colchón
dew point	temperatura de condensación/ temperatura de rocío
discharge factor	densidad de plastificación
drying hopper	tolva de secado
eject-on-the-fly	expulsión mientras el molde abre
ejector pins	botadores
ejector plates	platos de expulsión
fill time	tiempo de inyección
flash	rebaba
gate	bebedero
gate freeze	endurecimiento de bebederos
hold	empaque
hold pressure	presión de empaque
hold time	tiempo de empaque
hot runner	colada caliente
hot drop/hot tip	punta caliente
injection rate	flujo de llenado
injection screw	tornillo de inyección
injection speed	velocidad de llenado
jetting	chorreo
manifold	distribuidor
melt flow	flujo del fundido
melt flow number	índice de fluidez

melt pressure	presión del fundido
melt temperature	temperatura del fundido
mold protect	protección del cierre del molde
molecular weight	peso molecular
nozzle	boquilla
nozzle tip	punta de la boquilla
parting line	partición del molde
pellet	gránulo
plastic residence time	tiempo de residencia
recovery	plastificación
robot	brazo mecánico
runners	coladas
shear rate	cambio cortante/ velocidad cambiante
shear stress	esfuerzo cortante
shear thinning	licuar por fricción
shot size	volumen de llenado/volumen de la unidad de inyección
sprue	palo
sprue bushing	casquillo
stack mold	molde doble
stress	esfuerzo
suck-back	rechupe
tie bars	máquina con barras
tiebarless	máquina sin barras
transfer point	posición de transferencia
transfer pressure	presión de transferencia
valve gate	válvulas de bebederos
vents	ventosas

VI - Spanish Terms in English

anilla	check ring
barril	barrel
bebadero	gate
bomba	blower
boquilla	nozzle
botadores	ejector pins
brazo mecánico	robot
cambio cortante/ velocidad cambiante	shear rate
casquillo	sprue bushing
cavidad	cavity
chorreo	jetting
colada caliente	hot runner
coladas	runners
colchón	cushion
contrapresión	backpressure
de inyección a empaque	boost to hold
densidad de plastificación	discharge factor
distribuidor	manifold
empaque	hold
endurecimiento de bebederos	gate freeze
equipo de refrigeración	chiller
esfuerzo	stress
esfuerzo cortante	shear stress
expulsión mientras el molde abre	eject-on-the-fly
flujo de llenado	injection rate
flujo del fundido	melt flow
gránulo	pellet
índice de fluidez	melt flow number
licuar por fricción	shear thinning
máquina con barras	tie bars
máquina sin barras	tiebarless
molde doble	stack mold
noyos	cores
palo	sprue

partición del molde	parting line
pedazos fríos	cold slugs
peso molecular	molecular weight
plastificación	recovery
platos de expulsión	ejector plates
posición de transferencia	transfer point
presión de empaque	hold pressure
presión de transferencia	transfer pressure
presión del fundido	melt pressure
protección del cierre del molde	mold protect
punta caliente	hot drop/hot tip
punta de la boquilla	nozzle tip
rebaba	flash
rechupe	suck-back
temperatura de condensación/ temperatura de rocío	dew point
temperatura del fundido	melt temperature
tiempo de empaque	hold time
tiempo de inyección	fill time
tiempo de residencia	plastic residence time
tolva de secado	drying hopper
tornillo con barrera	barrier screw
tornillo de inyección	injection screw
tornillo sin fin	auger
válvulas de bebederos	valve gate
velocidad de llenado	injection speed
ventosas	vents
volumen de llenado/volumen de la unidad de inyección	shot size

VII - Operational Costs

2010 Machine operating cost per hour, with operator and profit margin (North American molders)										
North America Tonnage Range	<50	50-99	100-299	300-499	500-749	750-999	1000-1499	1500-1999	2000-2999	3000+
Average (US\$/hr)	\$33.31	\$35.24	\$41.92	\$52.13	\$68.14	\$83.22	\$110.28	\$119.95	\$181.68	\$230.00
Average (US\$/sec.)	\$0.009	\$0.010	\$0.012	\$0.014	\$0.019	\$0.023	\$0.031	\$0.033	\$0.050	\$0.064

Note: - this is a reference; consult your finance department for more exact values.
 - assume that a profit margin of 10 to 15% is included.

(Use only to estimate productivity gains, in US\$)

Note: The cost could be divided into three types, Basic, Optional, and Special.

Basic	Optional	Special
Depreciation	Robot	Class 8 Clean Room
Building	TCU	Inspection or QC
Interest	Packing equipment	Engineering assistance
Maintenance	Special injector; LIM, two colors, high speed, ...	Tooling support
Electricity	Crane	Material testing
Water	Quick mold change	Packaging and labeling equipment
Miscellaneous	Special screw	Special product handling
Labor	Additive Feeder	Mold storage
Marginal benefits		Mold maintenance
Inspection and QC		
Material		
Waste		
Secondary services		
Mold		
Overhead		
Earnings		

Bibliography

A. Brent Strong, 2000, "Plastics Materials and Processing", ISBN: 0-13-021626-7
Application Engineering, 1981, "Application Manual Water Group Products"
Douglas C. Montgomery, 2001, "Design and Analysis of Experiment", Arizona State University, ISBN 0-471-31649-0
Georg Menges and Paul Moren, 1993, "How to Make Injection Molds", Society of Plastics Engineers, ISBN 3-446-16305-0
Hansjurgen Saechting, 1992, "Saechting International Plastic Handbook", Society of Plastics Engineers, ISBN 3-46-14924-4
Jay Carender, 1997, "Injection Molding Reference Guide", Advance Process Engineering
John Bozzelli, Jan 1998, "Process Optimization and Setup-card Data Requirements", Cycleset
John Bozzelli, Jan. 1997, "How to Set First-Stage Pressure", Plastics World
Lawrence E. Nielsen and Robert F. Landel, 1994, "Mechanical Properties of Polymers and Composites", ISBN 0-8247-8964-4
Peter Kennedy, 1995 "Flow Analysis of Injection Molds", ISBN:1-56990-181-3
R. J. Young and P. A. Lovell, 1991, "Introduction to Polymers", ISBN: 0-412-30640-9
Shiro Matsuoka, 1992, "Relaxation Phenomena in Polymers", Society of Plastics Engineers
Thermoplastic Troubleshooting Guide, 2000, Ashland Distribution Company
Thomas Pyzdek and Roger W. Berger, 1992, "Quality Engineering Handbook", ISBN: 0-8247-8132-5
Tony Whealan and John Goff, 1994, "The Dynisco Injection Molders Hand Book", Dynisco Instruments
ULF W. Gedde, 1995, "Polymer Physics", ISBN 0-412-62640-3

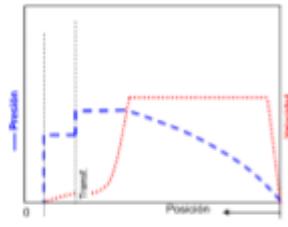
Answers

II. Injection Process Parameters

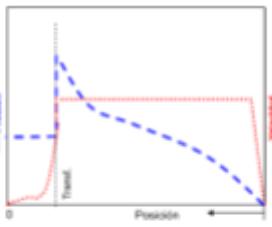
- 1) b. 2) b. 3) c. 4) b, c. 5) a. 6) b. 7) b, d. 8) a, d. 9) a. 10) b. 11) c, d. 12) b, c, d. 13) a. 14) b. 15) b. 16) b. 17) a. 18) b, c, d.

III. Process Graphs

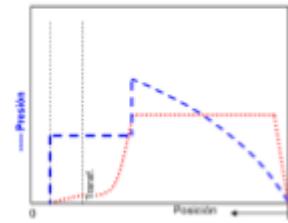
- 1) b. 2) b. 3) c. 4) b. 5) c. 6) a. 7) c. 8) c. 9) b.
10)



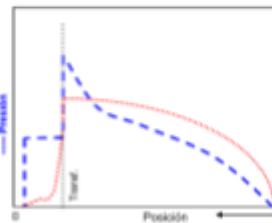
d) limited pressure



a) zero cushion



c) premature transfer



b) programmed speed was not reached

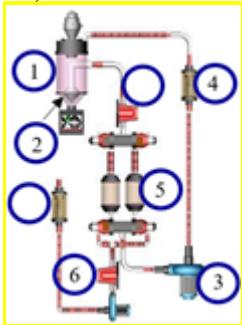
IV. Plastic Morphology

- 1) b. 2) c. 3) c. 4) a. 5) c. 6) c. 7) c.

V. Auxiliary Equipment

- 1) b. 2) c. 3) a, c, e. 4) b. 5) a. 6) b. 7) c. 8) b. 9) b. 10) a. 11) a. 12) b. 13) b. 14) a.

15)



Blending and Material Handling

- 1) b. 2) a.

Water Temperature Control to the Mold

- 1) b. 2) b. 3) c. 4) a. 5) b. 6) a.

VI. Molding from the Desk

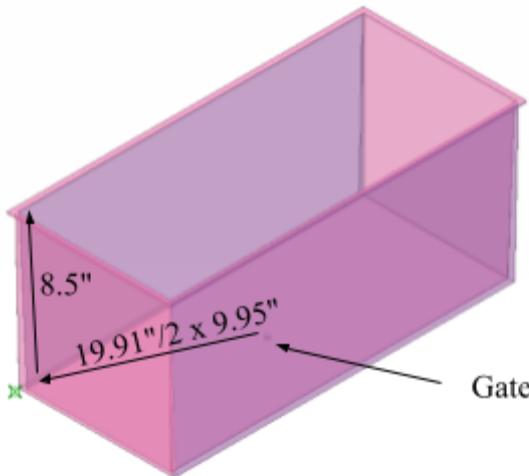
- 1) b. 2a) b. 2b) a. 2c) a. 3) d.

Injection Unit Sizing

- 1) b. 2) a. 3) a. 4) a. 5) a. 6) c. 7) a. 8) d.

9a) Area = 19.4in x 8.9in = **173 in²**

9b)



$$\text{Farthest flow path} = 8.5" + 9.95" = \mathbf{18.45"}$$

$$\text{Thinnest wall on the path} = \mathbf{0.08"}$$

$$\text{Thin Wall} - 18.45"/0.08" = \mathbf{230}$$

PF = 230 > 200; the force factor would be **2.5 ton/in²**

9c) Required clamping force = $173\text{in}^2 \times 2.5 \text{ ton/in}^2$
 $= \mathbf{433 \text{ US tons of force}}$ (1 ton = 2000 lb)

9d) Consumption = $1100/50\text{s} \times 3600\text{s}/1\text{h} \times 1 \text{ lb}/454 \text{ gr} = \mathbf{174 \text{ lb/hr}}$

9e) Required volume = $1100\text{gr}/0.92\text{gr/cc} = \mathbf{1196 \text{ cc}}$

9f) %U = $(1196\text{cc}/2480\text{cc}) * 100\% = \mathbf{48\%}$

9g) ton_{cooling} = $174 \text{ lb/hr} / 50 \text{ lb/hr/ton} = \mathbf{3.5 \text{ ton}_{cooling}}$

9h) gpm = $3.5 \text{ ton}_{cooling} \times 24 / 3^\circ\text{F} = \mathbf{28 \text{ gpm}}$

9i) %U = 48% and is between 35% and 65%.

The transfer would be between 12mm and 25mm.

$$\text{Transfer} = 25\text{mm} - 13\text{mm} (0.48-0.65)/0.3 = \mathbf{17.6\text{mm}} = \mathbf{0.69\text{in}}$$

9j) Recovery position = $1.27W/\delta D^2 + \text{Transfer}$
 $= 1.27 * 1100\text{gr}/[0.92\text{gr/cc} * (9\text{cm})^2] + 1.76\text{cm} = \mathbf{20.51\text{cm}} = \mathbf{8.07\text{in}}$

9k) Start with **700 psi (47 bars)** plastic pressure.

9l) $5\% \text{ of fill} = 0.05 * 8.07 \text{ in} = \mathbf{0.4 \text{ in}}$

9m) From the Material Data sheet = **410°F**

9n) From a data sheet of a generic PS:

Injection	Nominal Value	Unit
Rear Temperature	424 to 480	°F
Middle Temperature	424 to 480	°F
Front Temperature	390 to 415	°F
Nozzle Temperature	415 to 469	°F
Mold Temperature	60 to 150	°F

Since the $\%U$ is almost 50%, use the average.

Feed zone = Compression zone = Metering zone = **452°F**

$$\text{Nozzle} = (415^\circ\text{F} + 469^\circ\text{F})/2 = \mathbf{442^\circ\text{F}}$$

9o)

U%	T _r (# ciclos)
1%	140
2%	70
3%	47
4%	35
5%	28
6%	24
7%	20
8%	18
9%	16
10%	14
11%	13
12%	12
13%	11
14% - 15%	10
16% - 17%	9
18% - 19%	8
20% - 23%	7
24% - 27%	6
28% - 34%	5
35% - 46%	4
47% - 69%	3
>70%	2

Residence time (cycles) = **3 cycles**

Residence time(s) = 3 cycles x 50s/cycles = **150 seconds**

9p) Consumption = 174 lb/hr

Drying hopper volume = $174 \text{ lb/hr} \times 2 \text{ hours} / 35 \text{ lb/ft}^3$

$$= 9.94 \text{ ft}^3 = 281.5 \text{ liters}$$

9q) Dry air flow = 174lb/hr x 0.75 cfm/(lb/hr) = **130.5 cfm**

10) c.

VII. Machine Rheology

- 1) b. 2) c. 3) c. 4) b. 5) c. 6) a. 7) a.

VIII. Determination of Injection Speed

- 1) a. 2) b. 3) b. 4) (a) a. 5) b. 6) a, c. 7) b. 8) b, c. 9) c. **10) d.**

IX. Verifying Fill Balance

- 1) c. 2) b. 3) a. 4) a. 5) d. 6) b. 7) a. 8) b.

X. Determination of Hold Stage Parameters

- 1) a. 2) b. 3) b. 4) c. 5) a. 6) a, c. 7) b. 8) a. 9) c. **10) a.**

XI. Determination of Cooling Stage Parameters

- 1) b. 2) b. 3) d. 4) a. 5) a. 6) a. 7) d. 8) b.
9)

$$\frac{\overline{D}_C + \beta_0 T_M + \beta_1 t + \beta_2 T_F + \beta_3 T_M t + \beta_4 T_M T_F + \beta_5 t T_F + \beta_6 T_M T_F t}{(1) \quad (5) \quad (2) \quad (4)} \quad (3)$$

XII. Process Limits

- 1) a. 2) b. 3) a. 4) a. 5) b.

Index

A

additive feeder, 95
direct and volumetric, 95
gravimetric, 98
pneumatic proportional valve, 100
amorphous, 55, 145
approximated rheology, 16, 185, 213
by power, 179, 185, 188, 209
with viscosity, 181, 188

B

backpressure, 33, 153
barrel utilization (U%), 151, 282
blending, 95
blower, 79
bulk density, 76

C

calculations, injection unit, 142
central blending system, 101
check ring, 21
clamping force, 36, 123, 125, 280
combined effect, 59, 259
condensation temperature, 71
cooling, 22
parameters, 31
stage, 20, 51, 153, 249, 265
time, 31, 111, 235, 242, 251, 258
cushion, 31, 45, 153, 240
zero, 41, 48, 241

D

decompression, 34, 46
Delta T, 81, 106
density, 143, 150, 282
desiccant bed, 80
desplazamiento de inyección, 156, 284
dewpoint, 71, 83
dieseling, 125
dryers, 78
drying, 73
318

flow, 72
hopper, 74
material, 70
temperature, 71
time, 73, 76
drying systems, 83
central, 85
hopper above the extruder, 84
integrated, 85
portable, 84

E

ejectors, 36, 138, 140

F

feeding, 95
fill balance, 224, 227
fill limitations, 200
filter, 79
force factors, 123
fountain flow effect, 148

G

gate freeze, 20, 30

H

heat removed, 104
heat zones of the barrel, 164
heater, 81, 116
hold (pack), 20
parameters, 29
pressure, 235
time, 30, 242

I

injection, 18
flow, 27, 62, 150, 179, 285
parameters, 27
pressure limit, 42, 201
speed, 18, 42, 150, 207, 212

time, 65, 179, 191, 201, 207, 212
unit size, 142
volume, 28, 143, 179, 240
intensification ratio, 168

L

limited pressure, 47

M

material consumption, 103
maximum recovery, 158
melt flow, 148, 175, 225
minimum and maximum opening, 139

N

Newtonian flow, 178
nozzle tip, 144, 148

P

peak power, 179, 213, 284
platen spacing, 138
premature transfer, 47
press calculations, 122
press movements, 35
projected area, 124, 133
pseudoplastic, 178
PVT diagram, 50

Q

questions, 37, 52, 67, 92, 102, 120, 141,
170, 195, 220, 231, 246, 262

R

recovery, 22, 96, 153
parameters, 33
position, 41, 154, 283
speed, 157
zone, 46
regeneration, 81
residence time, 76, 158
resin loader, 78
runner, 20, 31, 126, 136
cold runners, 146

hot runners, 45, 158, 229, 244

S

screw flights, 158
semi-crystalline, 55
shear
rate, 177
stress, 176
thinning, 178
shot size, 142
specific density, 150, 282
sprue bushing, 144
stack mold, 137

T

TCU, 113
temperature profiles, 164
thermal imbalance, 229
thermoplastics, 55
thermosets, 55
thin wall calculation, 129, 281
three-plate mold, 136
transfer, 18
parameters, 28
position, 161
pressure, 184
troubleshooting, 271

U

Universal
data, 277
parameters, 26
Universal Molding™ (MU™), 15

V

viscosity, 177

W

water flow, 106

Z

zero cushion, 49

Expert Opinions

"Universal Molding™ is, in the Dominican Republic, an essential platform for the development of Dominican molders and the local industry has increasingly benefited by applying this knowledge to the improvement of its processes."

***Miguel Calcaño, Plastics Consultant
HDI Inc., Dominican Republic***

"One of Wallyco's greatest pleasures was always the research and professional development of young Puerto Ricans. If I'm not mistaken, it was in 2000 that we provided resin and machine time to Dr. Ivan Baiges' students; among those I remember Roberto Pastor. Days of more questions than answers, which laid the foundation for corroborating or denying stories of molders and understanding the science behind the technique. Yes science, not magic, black box, or dark art. Once the technique was verified, thanks to the help of people such as Drs. Gregorio Velez and Ivan, it was reduced to its minimum essence, and a verified process of the best practices was developed. It is written at the level of the user, the person who must improve the process without formal education. There have been many changes from 2000 to this day, which will continue as ***Universal Molding™*** still has much to discover and teach. It is a great pride for me to have been involved in its beginnings, to have used its processes and to have trained as an instructor, and I wish Hector and ***Universal Molding™*** to continue to help the plastics industry and all those young Puerto Ricans who want to better themselves and their homeland with their performance."

J. Wally Cruz, Engineer and Plastics Engineer Specialist

"Universal Molding™ is an excellent tool, not only for understanding the process of injection molding, but also for understanding the behavior of different types of plastics, in a simple way but always with a scientific basis. As an ***MU™*** student without any experience in the world of plastics, I was able to understand the groundwork and foundations of injection molding. Then, as an instructor, I could see how ***MU™*** helped so many people and industries optimize their molding processes with amazing results, not only in quality but also in economy."

***Laureano J. Rodríguez, Sr. Account Manager
West Contract Manufacturing***

"At the beginning of the 2000s, I was part of the revolution that was barely beginning in Costa Rica about how to scientifically establish a process during my time at Abbott Laboratories, which later became Hospira, and is known today as ICU Medical. It was there that the first **MUTM** exercise was done outside Puerto Rico, entirely in Spanish and, for the first time in Costa Rica, it showed a better way to obtain objective evidence about from where the validated parameters came in the injection molding process, making **MUTM** the pioneer of this revolution in CR. After that, the course was opened to other companies in the industry in Costa Rica, and it has been taught year after year to the present day. Subsequently, from 2008 to 2014 I had the privilege and pleasure as a member of HDI, Inc. to participate in seminars and conferences alongside Héctor Dilán as a presenter."

Harold Gamboa Calderón, Sr. Account Manager - Distribution PolyOne Corporation (Central America and Andean Region)

"I learned about **Universal MoldingTM** when I was just starting my professional career. Thanks to Héctor and the **Universal MoldingTM**, my learning curve in the field of injection molding was exponential. This gave me the necessary tools to apply science during the development of different molding processes and was my foundation for the future of my career in plastics engineering.

During those first steps with **Universal MoldingTM**, together with Héctor, we managed to develop Rheology by Power, which moved away from rheology by viscosity, but at the same time obtained specific results in less time. Rheology by Power helps us greatly optimize the injection stage in a simple, short, and accurate way.

Now, after about 15 years working in the injection molding industry, I can say that **Universal MoldingTM** is the basis and the most useful learning tool for anyone working in this area.

Héctor, thank you for the confidence and opportunity to work with you when I was just getting started in the industry."

***Billy Torres, Technical Services Manager
Microsystems UK***