Processing I

Control Utilization and Cycle Development

Tim Weston

Roboshot

Mosaic+

Endura Touch

Before we begin, a word about the study of Injection Molding

Lost? Very lost? If you don't know where you're going, any road will take you there! This book is intended as a roadmap for setup, troubleshooting, and problem-solving of the Injection Molding process. Injection Molding requires knowledge of polymer materials, molds, and tooling, machine control, auxiliary processes, as well as set up, validation and troubleshooting techniques.

Understanding the interaction of all of these contributing elements is essential to the ability to isolate, identify, and solve Injection Molding production problems. It is the key to becoming a profitable competitor in a world market.

If you don't know where you're going, any road will take you there.

- Lewis Carroll
"Alice in Wonderland"

Two Roadmaps? Injection Molding Process development, optimization, troubleshooting, and validation can be studied from an engineering approach or a scientific approach. The engineering view seeks to answer the question "how" while the scientific view tends to seek the answer to the question "why".

Roadmap 1 - The engineering view is the core of set up techniques such as decoupled molding, systematic molding, and interestingly enough, scientific molding. All of these techniques strive to provide an orderly approach to Injection Molding through experimentation, data collection, and analysis of the data. These techniques and the individuals who have contributed to their development have revolutionized the "art" of Injection Molding into a robust, structured approach to molding employed worldwide.

Roadmap 2 - The scientific view looks at the polymer (plastic) as it travels through the molding process from the pellet to part. Orientation, crystallinity, laminar flow, shear heating, fountain flow, and how a pellet melts on the screw are just a few of the concepts important in the scientific approach.

So which road to take? I believe that to be a true Master Molder problem solver it is important to master and understand both the engineering "how" and scientific "why". Some solutions lie on the engineering approach "roadmap" while others lie on the scientific approach "roadmap". The following pages are my attempt to reconcile the need for both engineering and scientific views of Injection Molding and to present these approaches as simply and clearly as possible.

I hope this helps you in your quest to become a true Master of Injection Molding and to reach your destination of being able to solve any Injection Molding problem.

Tim Weston

Some Information About You

- *Your Name*
- *Where you work*
- *What do you do?*
- *Years in Plastics Industry*
- *What would you like to learn about Injection Molding?*

Name	What would you like to learn?

Contents

Milacron

Processing 1

Control Utilization and Cycle Development

Day 1

Control Utilization

Decoupled Molding

Tuesday Control Utilization

7:30 a.m.	Check in with Refreshments
8:00 a.m.	Welcome
	- Introductions
	- Seminar Goals
	- Basic Terms
	- The Process
	- Goals of Injection Molding
9:45 a.m.	**Break/Networking**
10:00 a.m.	- Milacron Molding Machines
	- Roboshot Controls
	Preview the "Google Translate" Lab
	Lab Session 1: "Google Translate"
	Review Data from "Google Translate" Lab
	- Roboshot Data Collection
	- What is Decoupled Molding?
	- The Power of Decoupled Molding
	- How to Decouple a Cycle
Noon	**Lunch**
12:30 p.m.	- Velocity Controlled Fill
	- Types of Transfer
	- Setting the Dose, Shot Size to Transfer
	Preview The Short Shot Experiment
	Lab Session 2: The Short Shot Method
	Review Data from The Short Shot Experiment
2:30 p.m.	**Break/Networking**
2:45 p.m.	- Roboshot Transfer
	- Transfer on Position
	- Transfer on Time
	- Estimating the Dose using Universal Set Up
	- Pressure Hydraulic Machines
	- Pressure on Electric Machines
	- Intensification Ratio
	- Universal Set Up Examples
	- End of Day Wrap-Up with Q & A
5:00 p.m.	**Adjourn**

Wednesday Cycle Development

7:30 a.m.	Check in with Refreshments
8:00 a.m.	- Quick Review with Q & A
	- Is a Cycle Pressure Limited?
	Preview the Pressure Limited by fill time experiment
	Lab Session 3: Pressure Limited by fill time analysis
	Review Data from Pressure Limited by fill time
	- Delta P
	- Fill Pressure "On the Roof"
9:45 a.m.	**Break/Networking**
10:00 a.m.	- Using Graphs to Analyze Fill
	Preview Injection Pressure graph analysis
	Lab Session 3: Pressure Limited by IP graph analysis
	Review Data from Injection Pressure graph analysis
	- The Fill (95-98%) Part
	Preview Recoupling the Cycle
	Lab Session 4: Recoupling the Cycle
	Review Data from Recoupling the Cycle
Noon	**Lunch**
12:30 p.m.	- Orientation and Injection Velocity Optimization
	Preview of Injection Velocity Experiment
	Lab Session 5: The Injection Velocity Experiment
	Review Injection Velocity Optimization Data
	- Injection Velocity Profiling
	- What is a short shot series?
2:30 p.m.	**Break/Networking**
2:45 p.m.	Preview The Gate Seal Experiment
	Lab Session 6: Gate Seal Experiment
	Review Gate Seal Data
	- Amorphous and Crystalline Shrinkage
	- PvT Relationships
	Preview Optimizing Pack Pressure
	Lab Session 7: Pack Pressure, Maximizing C_{PK}
	Review Optimizing Pack Pressure
	- End of Day Wrap-Up with Q & A
5:00 p.m.	**Adjourn**

Thursday — Cycle Optimization

7:30 a.m.	Check in with Refreshments
8:00 a.m.	- Quick Review with Q & A
	- Runner Balance
	- Cooling Balance
	- a Balance of Fill Example
	Preview Balance of Fill analysis
	Lab Session 8: Mold Balance of Fill Parts
	Review Balance of Fill Data
9:45 a.m.	**Break/Networking**
10:00 a.m.	- Barrel Temperatures
	- Temperature Profiles
	Preview of Tacking Temperature Experiment
	Lab Session 9: Tacking Temperature
	Preview of Tacking Temperature Experiment
	Preview of 30-30-30 Melt Temperature Experiment
	Lab Session 10: The 30-30-30 Melt Temperature
	Review 30-30-30 Melt Temperature Data
	- Cooling, Delta T, and Ejection Temperature
Noon	**Lunch**
12:30 p.m.	- Tying it all together
	Cushion, Shot Size, Transfer, Suck Back
	Injection Fill Pressure Limit
	Injection Speed(s)
	Screw Speed, Backpressure, Barrel Heats
	Coolant Temperature and Flow Rate, Cooling Time
	Pack Pressure
	Hold Time
	Final Q&A
2:00 p.m.	**Adjourn**

GLOSSARY 101

Barrel - Machine component where resin is fed into for heating and mixing; contains the screw

Cavity - Hollow half of the mold; typically referenced as the A-side

Clamping Force - Force needed to keep mold closed while mold fills with plastic

Cold Runner - Channels in the mold that flow material to the part cavity; hardens with part and is removed after ejection

Cooling Channels - Channels built throughout the mold to control the mold temperature

Core - Half of the mold that fills the hollow side; typically referenced as the B-side

Cycle Time - Time it takes to complete one molded part; commonly measured from mold open to next mold open

Ejector Pins - Installed in the mold to eject the part from the mold when properly cooled

Flash - Excess material on molded part

Gate - Entry point for plastic into part cavity

Hot Manifold - Channel system that runs material to the part cavity and keeps material heated to avoid a runner

Knit Line - Occurs where different flow fronts of material meet while flowing through the mold

Mold Flow Analysis - Computer simulation of predicted material flow, fill, and warp in mold to assist in tool and part design

Mold (Tool) - Made of two halves (core and cavity); plastic is injected into mold and cooled to create molded part

Parting Line - Line formed in the molded part where two halves of the mold meet

Resin - Generic term for plastic material

Regrind - Material that has already been run through the machine but can be ground down and reused in a blend with virgin material

Screw - Within the barrel; moves the resin forward toward mold

Shot Size - Amount of plastic required to make part and runner

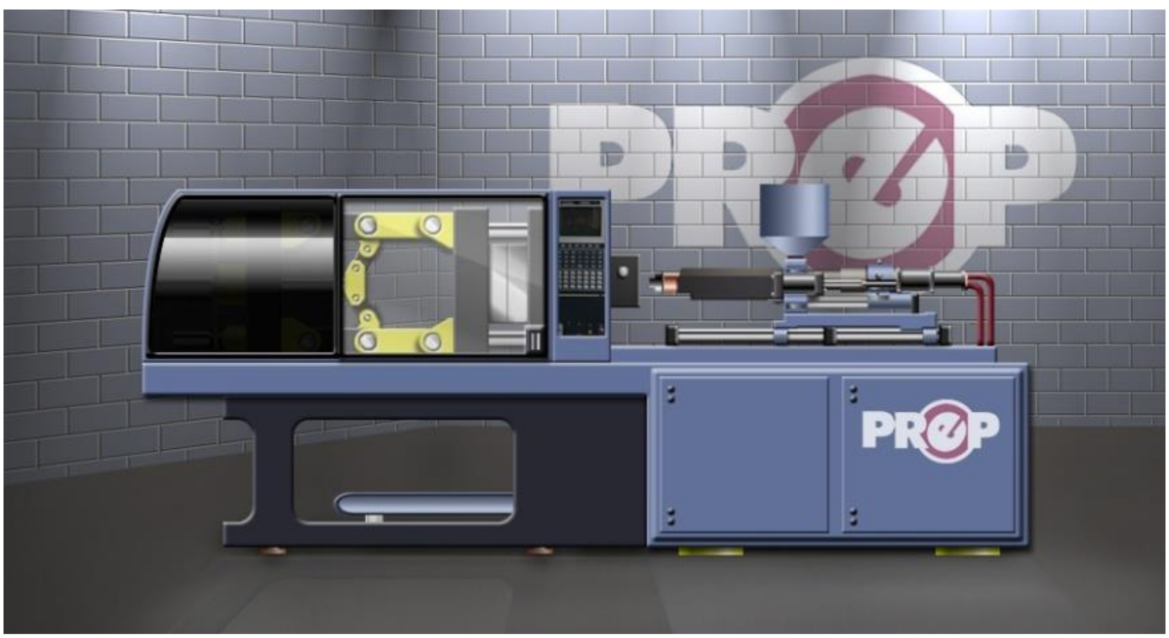
Sprue - Entry point for plastic into the mold from the barrel

Terminology – The Parts of an Injection Molding Cycle

Let's looking at the parts of an Injection Molding Cycle. Run the animation im.exe. Name as many parts of the cycle as you can as the animation runs. Note, as with the names of the parts of a machine, parts of the cycle may have different names based on "Tribal Knowledge"!

Start at:

1. **The mold closes.**
2.
3.
4.
5.
6.
7.
8.
9.
10.
11.
12.



(the answers for this exercise are located at the back of the book)

Terminology – The Injection Molding Cycle / Sequence of Events

Let's look at the parts of a cycle another way. Let's look at when each part of a cycle occurs for the Injection End and the Clamp End. The diagram illustrates the sequence of events for both the Injection End and Clamp End. The sequence shown is for a setup technique known as decoupled molding. This specific sequence is decoupled 2 setup. We will discuss this setup technique in great detail later.

Fill to Pack Transfer >
(aka: V/P transfer)

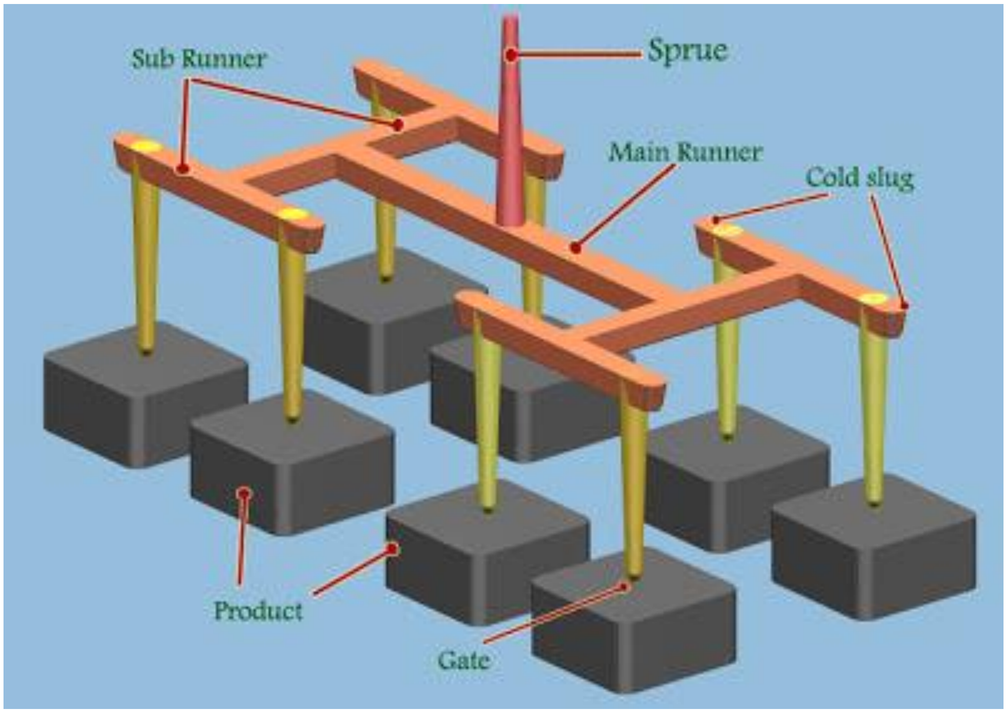
<table>
<tr><td>Injection End</td><td>Wait</td><td>Inj. Unit Forward</td><td colspan="2">Inject Fill</td><td>Inject Pack/Hold</td><td>Make next shot</td><td>Suck Back</td><td>Inj. Unit Back</td><td colspan="4">Wait</td></tr>
<tr><td>Clamp End</td><td>Mold Closes</td><td colspan="2">Wait</td><td colspan="5">Part Cools</td><td>Mold Opens</td><td>Cores Out</td><td>Eject Part</td><td>Cores In</td></tr>
</table>

<u>Terminology – Cycle Terms (Pressures, Times, and Distances)</u>

Fill:	The part of the cycle where the mold is filled with plastic.
Pack:	The part of the cycle where the plastic is compressed under high pressure.
Hold:	The time the pack pressure is applied.
Pack and Hold:	A part of the molding cycle. Pack and Hold work together. For example, a pack pressure for a hold time. Aka: Second Stage.
Pack Pressure	The pressure the machine uses during the pack and hold portion of the cycle.
Hold Time	The time the pack pressure is applied.
First Stage:	Another name for the fill part of the cycle.
Second Stage:	Another name for the Pack and Hold part of the cycle.
The Shot	A melted pool of plastic in front of the screw.
Shot Size	The screw position reached to build the shot.
Transfer	The screw position where the machine switches from reading fill settings to pack and hold settings.
The Dose	The fill volume. It is the distance from Shot Size to Transfer.
The Cushion	The melted plastic in the shot left after fill, pack, and hold.
Injection Speeds	The speeds the screw uses during the fill portion of the cycle.
Fill Pressure Limit	The maximum pressure allowed for the screw to reach Injection Speeds during the fill portion of the cycle. aka: First Stage
Cavity Pressure	The pressure in the cavity during fill, pack, and hold.
Suck Back	Pulling the screw back after the shot has been made. Aka: Melt Decompression.
Sprue Break	Moving the Injection Unit away from the mold on each cycle.

Terminology – The Molded Shot

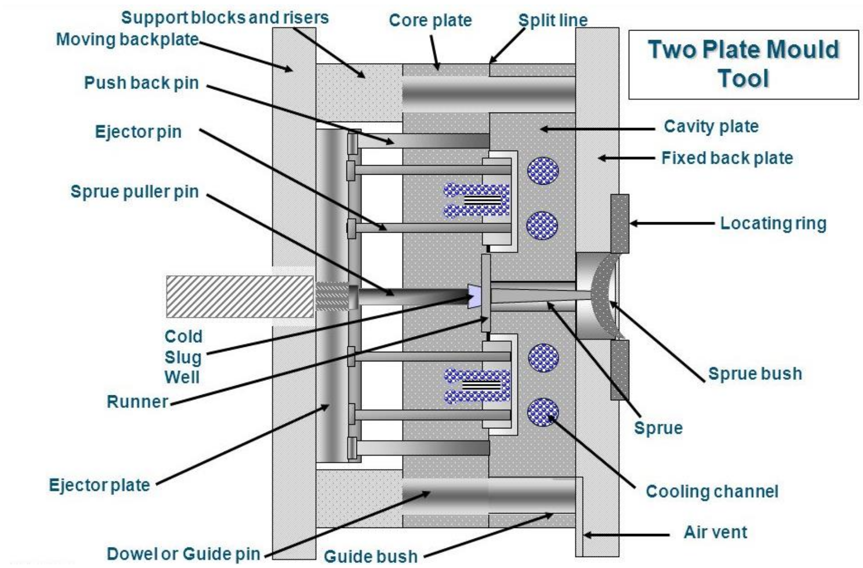
When plastic material is injected into an empty mold the plastic fills all the empty spaces. This produces the desired product in the cavities but also fills a system of channels used to reach each cavity called a runner system. The runner system can be broken into the sprue runner segments, gates, and the molded parts.



Molded Shot:	The complete cooled plastic part molded each cycle.
Part:	The part of the molded shot produced to sell.
Cavity:	Part of the mold where the part is formed.
Runner:	The part of the molded shot used to deliver the shot to the cavity. Also, the part of the mold that forms the molded runner.
Parting Line:	The plane where the two mold halves separate.
Sprue:	Part of the molded shot formed between the machine nozzle and the parting line.
Runner Segments:	Part of the molded shot formed along the parting line.
Primary Runner:	Section of the runner attached to the sprue.
Secondary Runner:	Section of the runner formed after the first turn in a runner.
Tertiary Runner:	Section of the runner formed after the second turn in a runner.
Gate:	A small cross-section of the runner found between the main runner and the part.
Shrinkage	The change in dimensions observed when plastic parts cool.

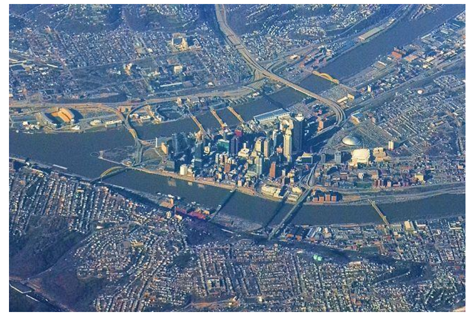
Terminology – Parts of and Injection Mold

The molded shot is formed in an Injection Mold. Molds are typically made of high strength steel. Molds are designed to support many tons of crushing force and many thousand PSI of plastics pressure. The illustration below names the parts of a common mold. Like all the other terminology we studied, special molds will have additional components, or the components might be named slightly different.



Mold "A" Side:	The half of the mold attached to the fixed (stationary) platen.
Mold "B" Side:	The half of the mold attached to the moving platen.
Locating Ring:	A ring that fits into the mounting hole in the fixed platen.
Sprue Bushing:	An insert that passed through the Cavity Plate. Forms the sprue.
Front Clamp Plate:	The mold plate closest to the fixed platen.
Rear Clamp Plate:	The mold plate closest to the moving platen.
Cavity:	The mold component that forms the outer surface of the part.
Cavity Plate:	The plate that holds the cavities. aka: The "A" plate.
Core:	The mold component that forms the inner surface of the part.
Core Plate:	The plate that holds the cores. aka: The "B" plate.
Leader Pins:	A stout pin that aligns the mold halves. aka: Guide pins.
Leader Pin Bushing	A bushing the leader pin fits into.
Sprue Puller Pin	A pin with an undercut that helps pulls the molded sprue.
Ejector Pins	Pins used to push the molded part from the cores.
Ejector Plate	The mold plate that drives the ejector pins.
Ejector Retainer	The mold plate that holds the ejector pins in place.
Coolant Lines	Channels in the mold for coolant.

Keys to Understanding the Injection Molding Industry



Let's begin our study of the Injection Molding Industry with an overview of the industry and some key elements to why it is such an important manufacturing process.

1. Companies are in business to make money.

First and foremost, companies are in business to make money. Each company doing injection molding only uses injection molding to make a part because it is the most profitable way to make that part. Profit isn't a dirty word. It keeps investors happy, allows companies to pay higher wages, provides more time off, gives better benefits, and a cleaner safer work environment. Lowering the cost of molding a part also makes the customer for that part more competitive. They sell more parts, buy more parts from you, and make you even more profitable. Everyone wins (except your competitors). The owner of the first company I worked for had a saying. "Business is like war. The only difference is the losers in war go to a cemetery and the losers in business go to the unemployment office". That saying is a bit crude but certainly conveys the right idea. Everything we say in the coming pages will be colored by the need for profitability.

Injection Molding is a very efficient way to make large numbers of parts, both simple a very complex. Below are two examples that will illustrate this.

<u>Activity 1- Keep an Eye on the Money</u>

The bottle cap at the right is a well-known product. Calculate the total value of parts produced in a year given the following manufacturing data:

Plastic Temperature: 450°F
Part Ejection Temperature: 150°F
Cooling Time: 15 seconds
Cycle time: 20 seconds
Mold Cavities: 64 (parts per cycle)
Net $/Part over material cost: $0.06 (six cents)
Assume: 24 / 7 operation. 50 weeks a year.
??? = Annual Total Value ($) of Parts

Answer: $5,806,080

If the plastic temperature could be reduced to 420°F, the cooling time (to 150°F Ejection Temperature) could also be reduced, let's say to 12 seconds. How much more would the total value of the parts be, produced in a year?

Answer: $6,830,682 (an extra $1,024,602 !!!)
Wow! Cha-Ching, a cool million extra. Same equipment, same mold, same floor space, etc. Nothing different except a little cycle change.

$ $ $

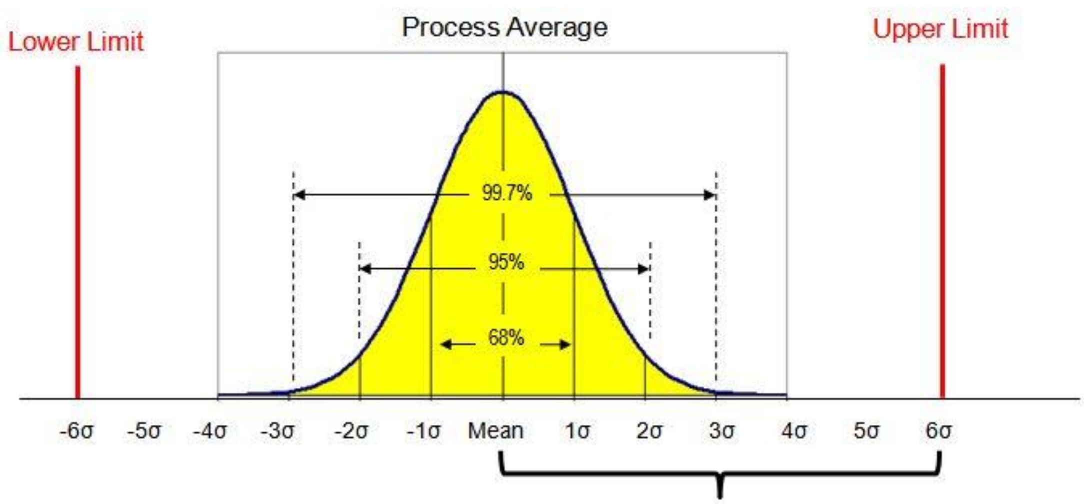
2. Customers don’t pay for bad parts.

The examples in Activity 1 demonstrate the amazing manufacturing power of injection molding. Once a mold is built, millions and millions of parts can be reproduced from the mold cavities. There are 110 million parts produced in the 17-second scenario with a value of $6,830,682 over the raw material cost (value-added).

There is only one problem with such a prolific manufacturing process. The customer will only pay for parts that meet specifications. **Customers don’t pay for bad parts.** Injection molding is capable of producing bad parts just as quickly as it can produce good parts! In fact, you can produce bad parts faster than you can produce good parts. (not sure why you would want to?)

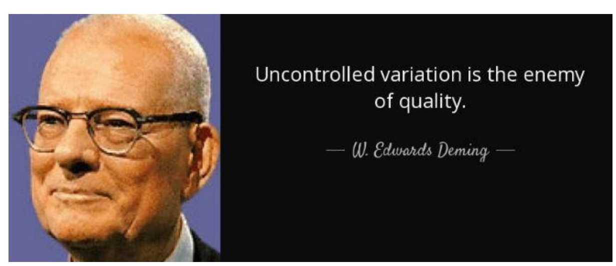
“Variation is the Enemy”

If we could <u>always</u> produce identical parts from <u>identical cavities</u> in Injection Molding, there would <u>never</u> be bad parts! Unfortunately, natural, or common causes of variation are present in any manufacturing process. The constant battle is to eliminate assignable causes of variation and keep the parts produced between the customers lower and upper specification limits. In the battle to produce quality parts, “Variation is the Enemy”.



Quality and quality concepts, like six sigma, Cpk, lean, and total quality, are an important part of mastering the Injection Molding process. Those concepts are covered in detail later.

“Variation is the Enemy” is not an original idea here. Famed quality guru W. Edwards Deming originated this idea.



3. Protect the Investment!

An Injection Molding Machine costs about $1,000/ ton of clamping force. An Injection Mold often costs about the same as the Injection Molding machine it runs in.

For the example in Activity 1, let’s assume we needed a 500-ton machine. Using the estimate above that machine should cost about $500,000. Add a mold of equal value, another $500,000, and our total investment in just the machine and mold is $1,000,000. (that’s right, 1 million dollars).

Notice in Activity 1 we chose not to save cycle time by closing the mold faster. That might slam the mold closed and damage the mold and machine over time. The time saved would produce a more profitable operation but at the expense of destroying the million-dollar investment in mold and machine! Wise idea?

4. Save a Penny whenever you can.

The last key to understanding the Injection Molding industry is that it is good to save a penny whenever you can. There are many little things, and some big things too, that can save lots of “pennies” in the Injection Molding business, In Activity 1 we looked at how lowering the plastic melt temperature a few degrees lowered cooling time and cycle time producing higher profit. As it turns out, lowering the plastic temperature also saves on the electricity it takes to heat the plastic from room temperature to the plastic melt temperature.

Milacron Molding Machines / Control Systems

<u>ALL ELECTRIC</u>

Roboshot
Precise 31-iB Control

PowerPAK
Toggle
Mosaic+ Control

<u>Servo</u>

C-SERIES
2 Platen Technology
Mosaic+ Control

MAXIMA PERFORMANCE
2-platen
Mosaic+ Control

MAGNA TOGGLE
Toggle
Endura Touch

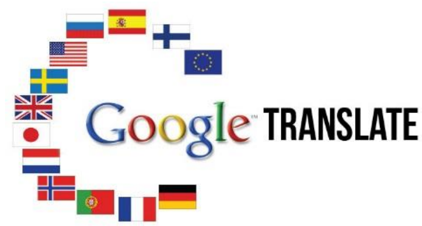
Q-SERIES
Toggle
Endura Touch

Cycle Sheets, Process Settings

ACME MOLDING COMPANY				
MACHINE: (#16) 275 Ton α-S275iA				
MOLD: 16 cavity Mold-Masters Hot Runner with Valve Gate				
MATERIAL: High Impact Polystyrene		COLORANT: Chroma Color FDA P/S White		
CUSTOMER: Wil E. Coyote		PART: Roadrunner Trap		
	Molding Parameter	**Unit**	**Value**	**Machine Capability**
1	Melt Temperature	°F		
2	Nozzle Temperature	°F		
3	Metering Zone Temperature	°F		
4	Compression Zone Temperature	°F		
5	Feed Zone Temperature	°F		
6	Screw Speed	RPM		
7	Backpressure	PSI		
8	Injection Time (Fill Time)	sec.		
9	Cycle Time	sec.		
10	Injection Pressure	PSI		
11	Injection Speed(s)	in./sec.		
12	Pack Pressure(s)	PSI		
13	Hold Time	sec.		
14	Shot Size (Screw Position)	in.		
15	Transfer Position	in.		
16	Desired Cushion	in.		
17	Mold Temperature	°F		
18	Circulator Temperature “A” side	°F		
19	Coolant Flow Rate “A” side	gal/min		
20	Circulator Temperature “A” side	°F		
21	Measured Steel Temperature “A”	°F		
22	Delta T “A” side	°F		
23	Reynolds Number “A” side			
24	Circulator Temperature “B” side	°F		
25	Coolant Flow Rate “B” side	gal/min		
26	Measured Steel Temperature “B”	°F		
27	Delta T “B” side	°F		
28	Reynolds Number “B” side			
29	Clamping Force Needed	US Tons		
30	Dryer Set Temperature	°F		
31	Drying Time	hr.		
32	Dryer Throughput	lb./hr.		
33	Granulator ID			S Cutter, low speed granulator
33	Nominal Part Weight	g.		

Translating the Controller

Each Injection Molding machine does the same basic functions as we discussed previously. The actual control settings and computer logic used to control these basic functions vary widely among machine manufacturers. For every Injection Molding machine, it is necessary to “translate” the settings on a machine to the standard terminology we have just discussed. The following is a list of key settings we will need to know to apply the theory of Decoupled molding any machine. It would be great if Google Translate work for Injection Molding machines!



<u>Machine Settings – Settings we will need to know</u>

Determine how to:

Navigate within the Control Software.

Enter Data (make changes) in data fields.

Determine the location of the following machine control settings:

Injection Pressure(s)

Injection Speed(s) (Injection Velocities)

Transfer program (ie: time, position, pressure, etc.)

Transfer Position

Pack Pressure

Hold Time

Also find:

Cooling Time

Shot Size

Determine how to monitor:

Actual Fill Time

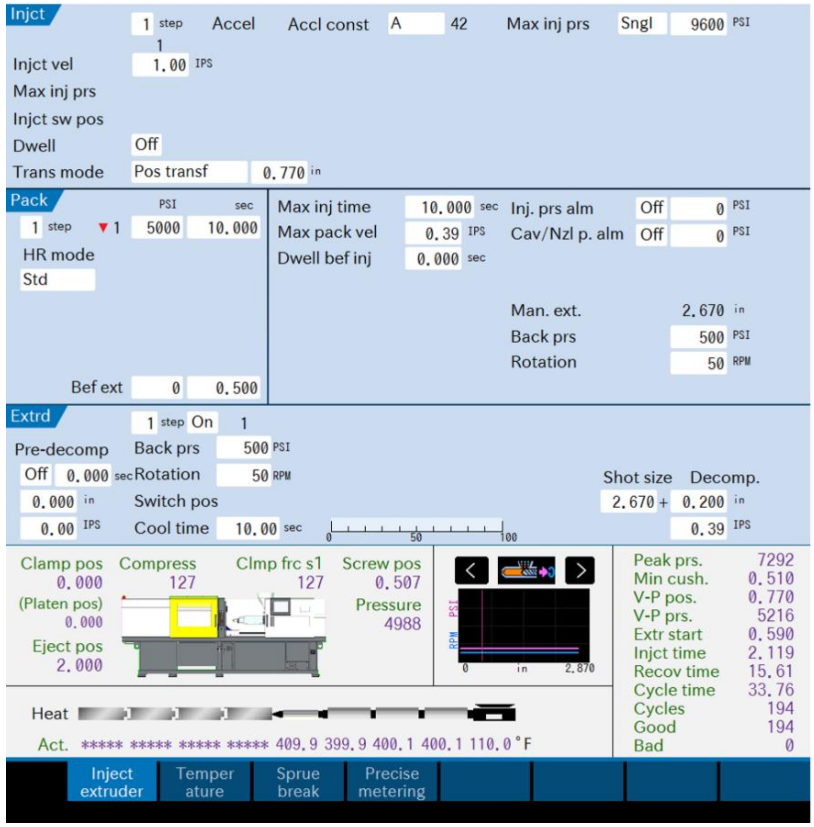
Actual Pressure at Transfer

The Cushion

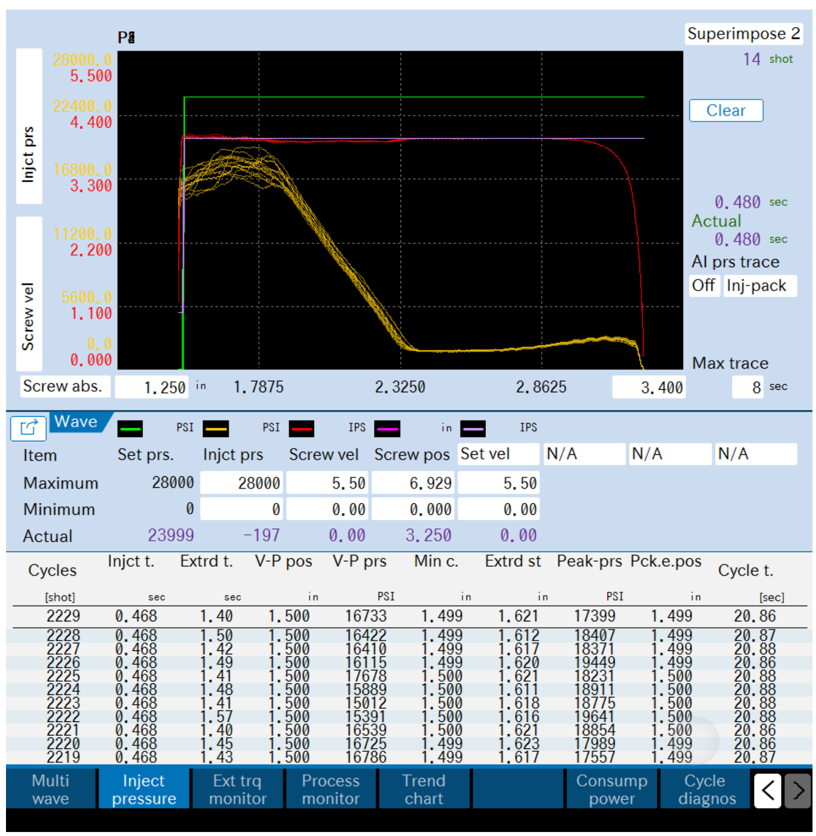
Determine the Machine Intensification Ratio

Roboshot Controls

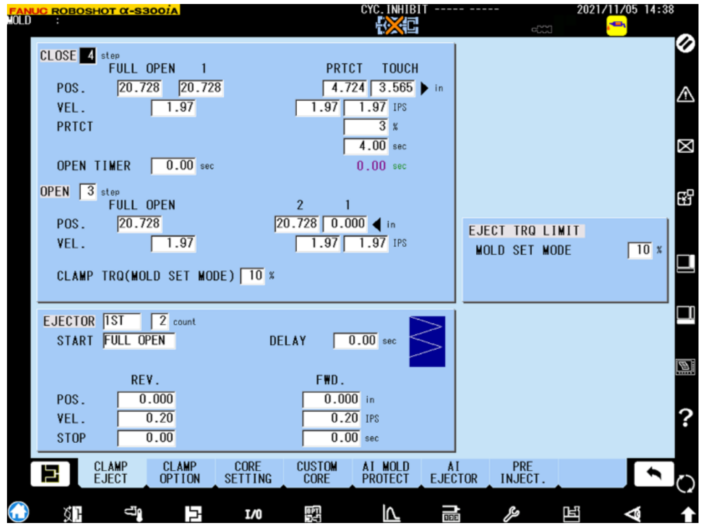
Inject / Extruder



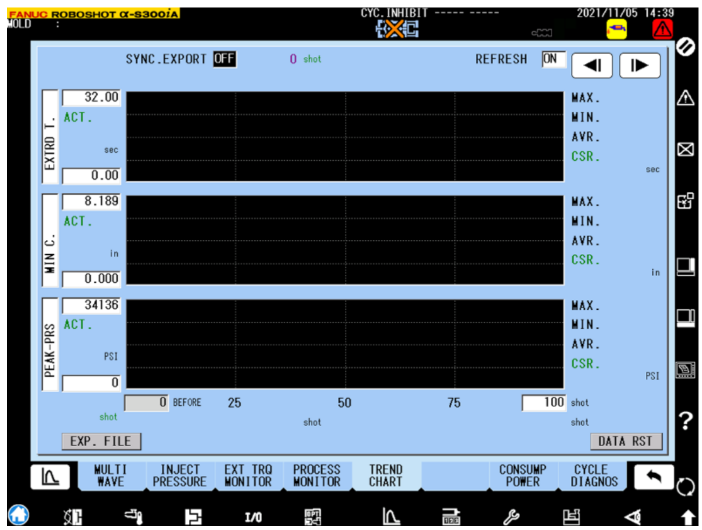
Inject Pressure Graph



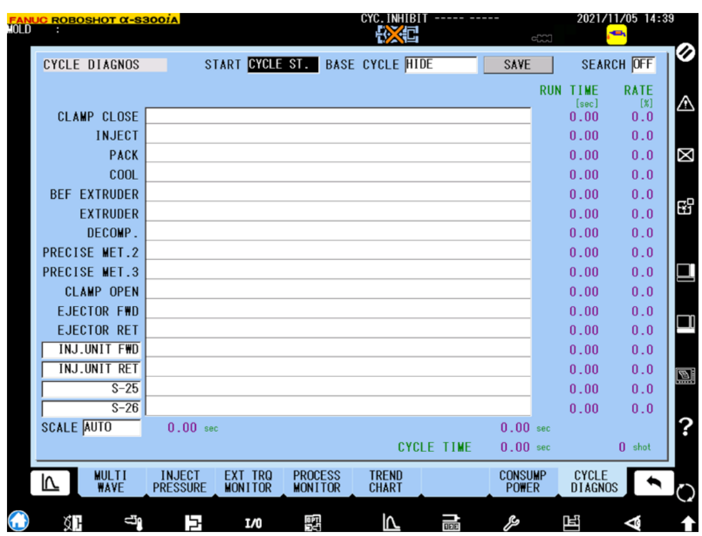
Clamp Setup



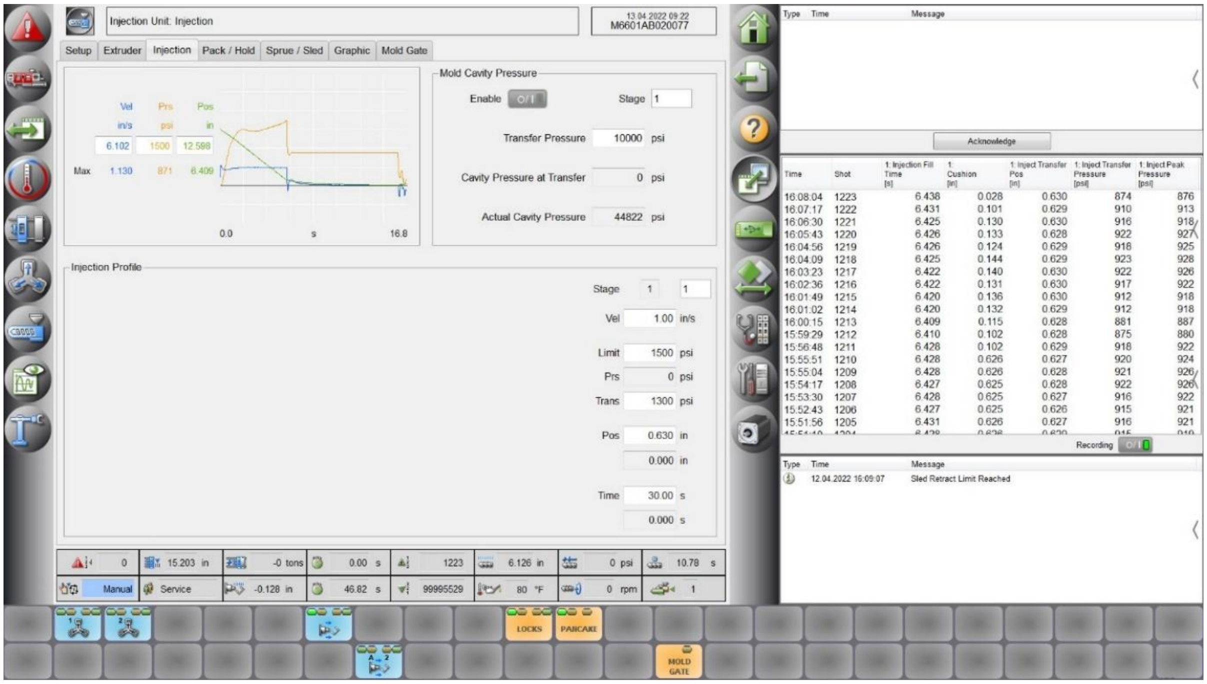
Clamp Open/Close

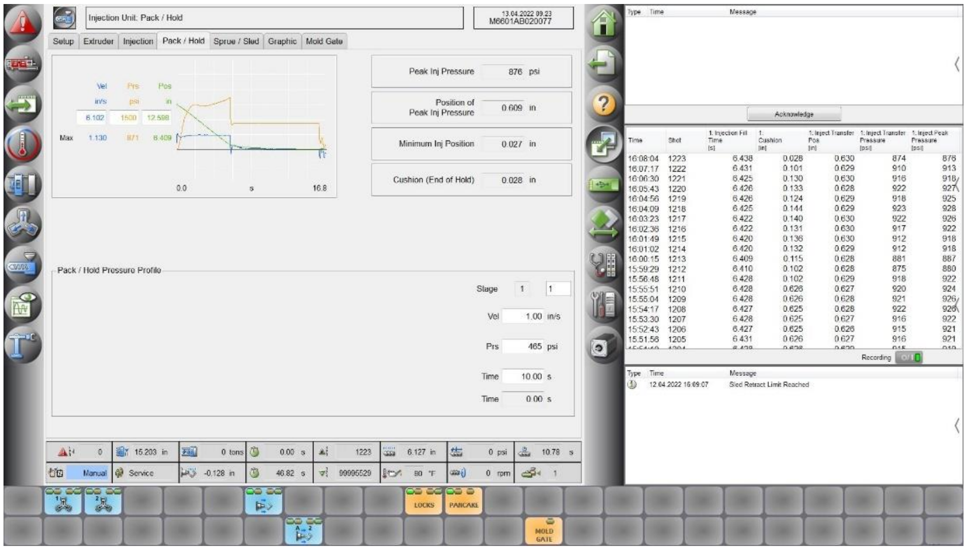


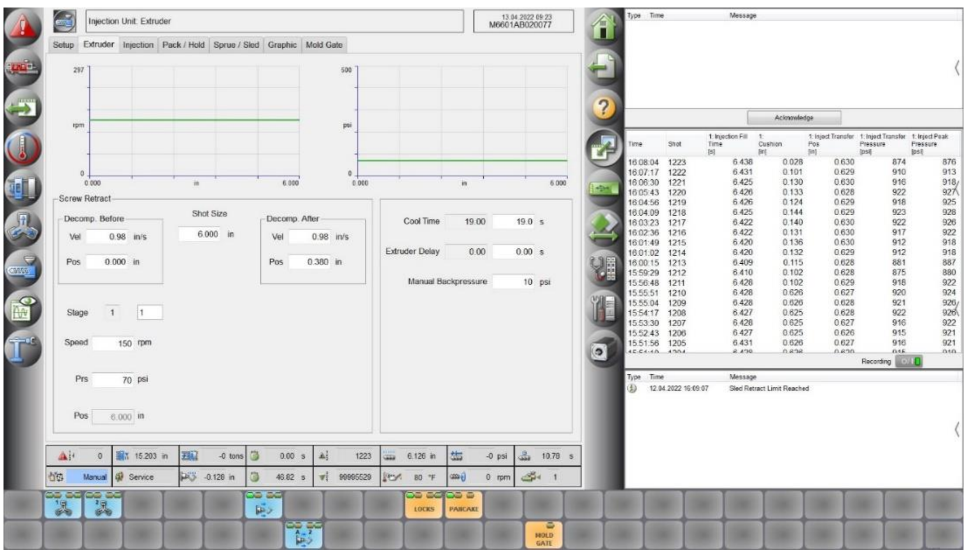
Clamp Ejector



Mosaic+ Controls (Endura Touch Logic)




Injection Unit: Pack / Hold
Setup Extruder Injection Pack / Hold Sprue / Sled Graphic Mold Gate
Peak Inj Pressure 876 psi
Position of Peak Inj Pressure 0.609 in
Minimum Inj Position 0.027 in
Cushion (End of Hold) 0.028 in
Pack / Hold Pressure Profile
Acknowledge
Sled Retract Limit Reached


Injection Unit: Extruder
Setup Extruder Injection Pack / Hold Sprue / Sled Graphic Mold Gate
Screw Retract
Decomp. Before
Shot Size
Decomp. After
Cool Time
Extruder Delay
Manual Backpressure
Acknowledge
Sled Retract Limit Reached


Temperatures: Overview
Overview Barrel Heating Adjustment
Temperature Reduction
Barrel Release
Oil Temperature
Oil Preheat
Acknowledge
Sled Retract Limit Reached

Technician or Firefighter? It's up to you!

This is a Molder:

This is a Firefighter:

A Molder is not a Firefighter!

(or at least they shouldn't be!)

The Processor's Dilemma

Part of the difficulty in learning to control the injection molding process centers around the observation that there is not a direct cause and effect relationship between machine settings and molded part properties.

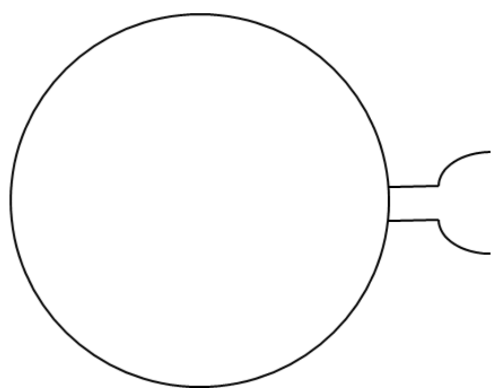
For years molders have struggled to tie part property problems to specific machine settings. The connection between part properties and machine setting can only be understood when we account for some fundamental material variables and their relationship to four basic process variables.
The following figure illustrates these relationships.

Machine Settings		Part Properties
*Injection Pressure *Injection Speed *Melt Temperature *Mold Temperature *Transfer *Circulator Temperature *Flow Meter Settings	⟷	*Dimensions *Attributes *Mechanical Properties *Physical Properties *Other Performance Properties

1) The outside diameter of round parts being molded is measuring lower than the customer specification (the parts are undersized). Which machine setting would you pick to solve this problem?

2) A part has intermittent parting line flash. Which machine setting would you pick to solve this problem?

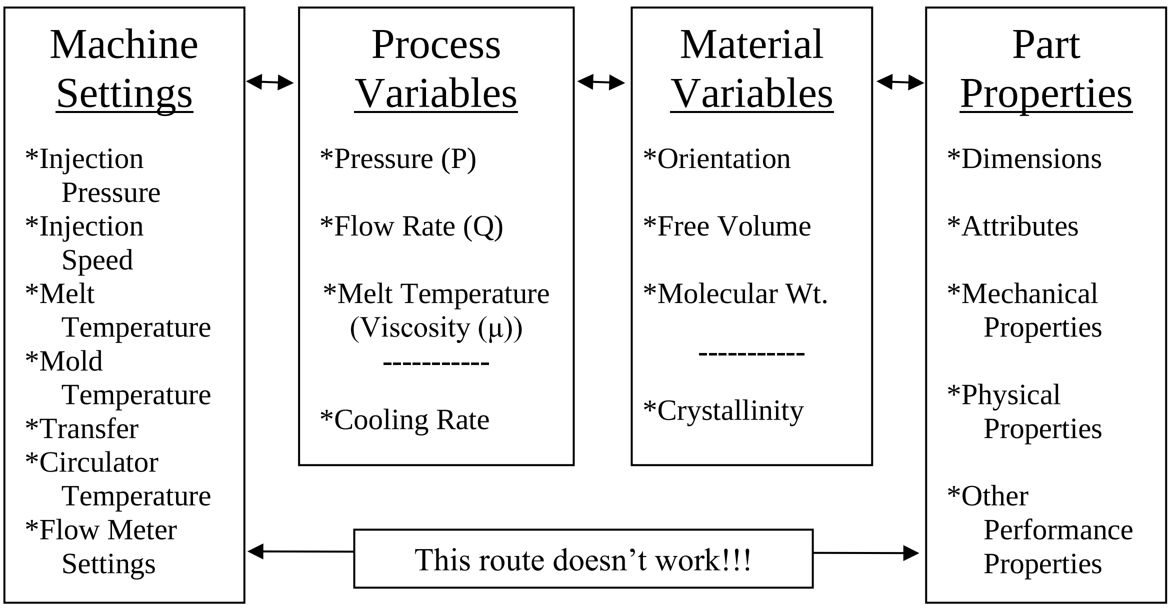
3) Parts are short. Which machine setting would you pick to solve this problem?



Single and Multiple Variable Cause and Effect

Part of the difficulty in learning to control the injection molding process centers around the observation that there is not a direct cause and effect relationship between machine settings and molded part properties.

For years molders have struggled to tie part property problems to specific machine settings. The connection between part properties and machine setting can only be understood when we account for some fundamental material variables and their relationship to four basic process variables.
The following figure illustrates these relationships.



We will use this figure to optimize and troubleshoot single variable and multiple variable injection molding problems.

The phrase "single variable cause and effect" is used to describe the causal relationship between any two things. One example of "single variable cause and effect" would be "when the sun comes up it will get light outside". One result is directly driven by one action. In Injection Molding we would like to define these one-to-one single variable relationships, for example increasing injection pressure will increase part size (through reduced shrinkage).

As you might guess, "multiple variable cause and effect" or "multivariable cause and effect" refers to a causal relationship where changing one input variable may produce more than one outcome. One example might be "if I drink a beer, it will help quench my thirst and will cause me to gain weight". Quenching my thirst was an intended outcome of drinking a beer. Gaining weight was not an intended outcome but happens anyway.

Frequently in Injection Molding, changing one process input changes an intended process outcome variable but also changes other possibly unintended process outcome variables

The Critical Process Valuables in Injection Molding

Process Variables
*Pressure (P)
*Flow Rate (Q)
*Melt Temperature (Viscosity (μ))
----
*Cooling Rate

The four critical process variables for Injection Molding are shown at the left. If these four variables are consistent then consistent parts will be produced. Remember; "Variation is the Enemy". Consistency is how variation is controlled and defeated. Let's look at each of the four critical process variables individually.

Flow / Fill Rate (Control Data - Fill Time)

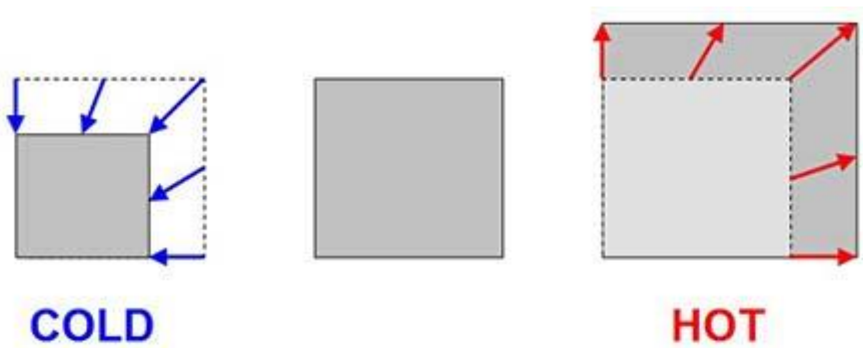
Flow rate directly determines the amount of orientation developed during fill for any given cycle. The amount and location of this developed orientation is determined by the cooling rate. Flow rate also impacts the plastic temperature as the mold fills. Frictional heating as the plastic flows increases with increased flow rate.

Plastic Melt Temperature (Control Data – Air shot melt temperature)

Plastic Melt Temperature before injection controls the viscosity (thickness) of the melted plastic. Heat from barrel zone heater bands and screw rotation during melt development both impact the temperature (and viscosity) of the melted plastic. The molecular weight of the plastic (melt index) also impacts the observed viscosity.

Pressure / Cavity Pressure (Control Data - Part Dimensions and Part Weight)

Pressure is used to squeeze the melted plastic to control the amount of contraction (shrinkage) when the part cools. More pressure produces less shrinkage and larger part dimensions.

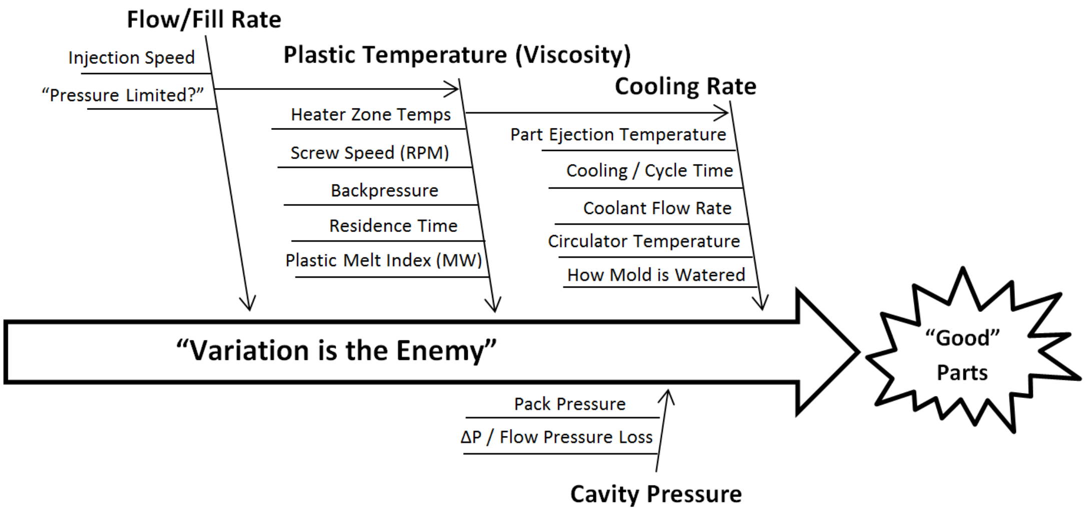


Cooling Rate (Control Data - Melt Temperature and Temperature at Ejection)

The rate of cooling determines how much of the developed properties are retained. Rapid cooling (quenching) freezes in more of the properties developed during fill, pack, and hold. Slow cooling (annealing) has the opposite effect, less of the properties developed during fill, pack, and hold are retained. Cooling Rate also controls the amount of internal molded-in stress and the amount of differential shrinkage (warp and anisotropy).

Sources of Variation in Injection Molding

The Ishikawa fishbone diagram below illustrates the four critical process variables and the machine, mold, and material settings associated with each. Some of these relationships are nice single variable relationships. For example, increasing pack pressure reduces shrinkage. Other relationships are multivariable. For example, increasing injection speed (fill rate) increases the amount of orientation produced **<u>and</u>** reduces the viscosity of the plastic!



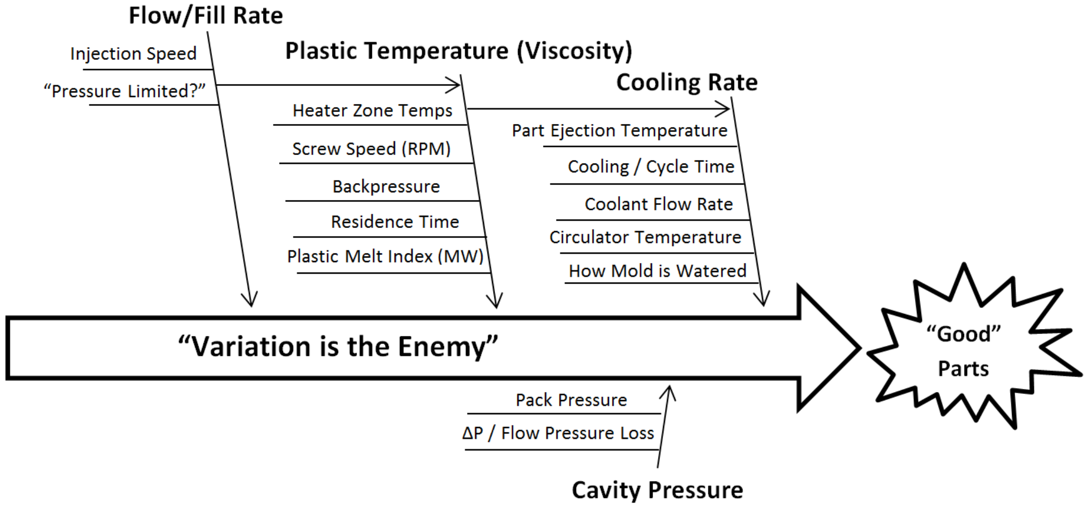
"Variation is the Enemy"

We need a **plan.** We must **do** the plan. The plan must include data to **check** the plan to eliminate variation in an out-of-control process. This plan is <u>Decoupled Molding</u>.

This is how to slay the dragon!

Controlling Variation using Decoupled Molding

The Ishikawa fishbone diagram below illustrates the four critical process variables and the machine, mold, and material settings associated with each. Some of these relationships are nice single variable relationships. For example, increasing pack pressure reduces shrinkage. Other relationships are multivariable. For example, increasing injection speed (fill rate) increases the amount of orientation produced **and** reduces the viscosity of the plastic!



Decoupled Molding uses a series of systematic experiments to determine the optimum setting for each process variable. Those experiments are:

Short Shot Experiment
- Shot Size
- Transfer Position
- Cushion

Velocity Optimization Experiment
- Injection Speed

Gate Seal Experiment
- Hold Time

Ejection Temperature
- Mold Temperature
- Coolant Flow Rate
- Cooling Time

Maximizing C_{PK}
- Pack Pressure

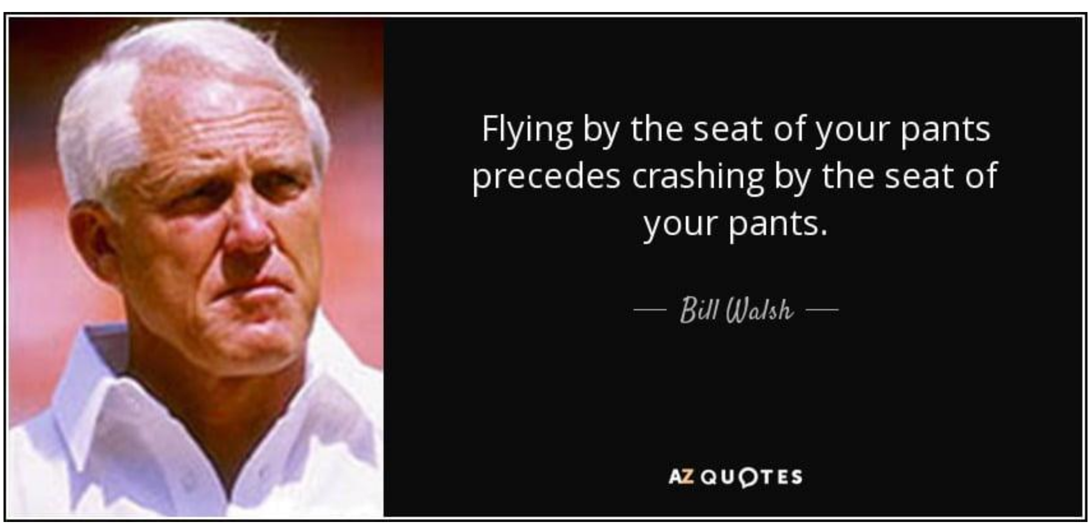
Tacking Temperature Experiment
- Screw Speed
- Backpressure
- Barrel Heats

Pressure Limited Experiment
- Injection Fill Pressure Limit

Air Shot Melt Temperature
- Melt Temperature

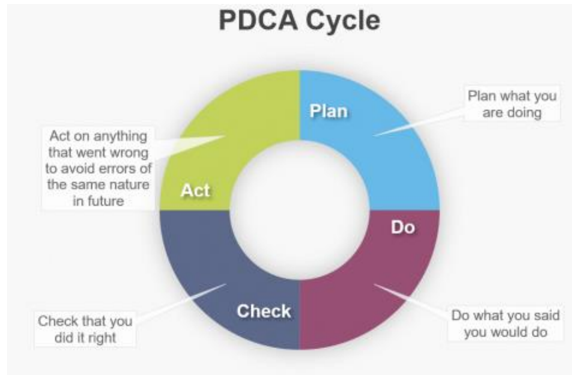
Balance of Fill Experiment
- Multi-cavity Consistency

Processing by Design (We need a plan!)



The Plan

Do, Control the Melt Temperature
- Set Zone Temps
- Set Screw Speed
- Set Back Pressure

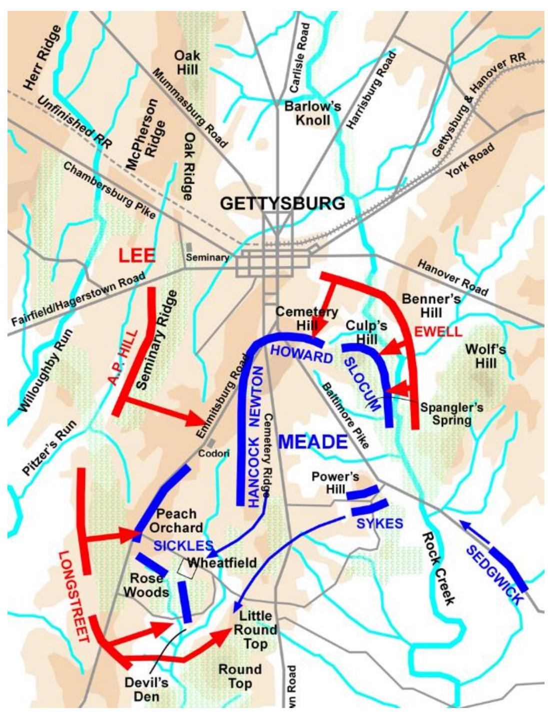


Do, Control the Mold Temperature
- Set Flow Rates
- Set TCU Temperatures

Do; Decouple
- Zero Pack Pressure
- Zero Hold Time

Do, Build a Data Driven Filling Phase
- Set Shot Size and Transfer, Short Shot Experiment
- Set the Optimum Injection Speed, Velocity Optimization Experiment
- Set the Injection Pressure, Pressure Limited Experiment

Do, Recouple
- Add an arbitrary Hold Time
- Add an arbitrary Pack Pressure



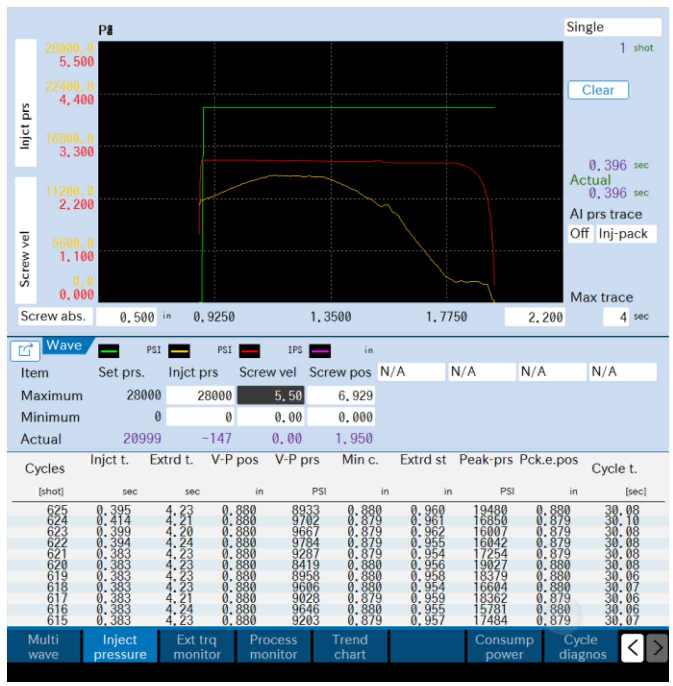
Do, Build a Data Driven Pack and Hold Phase
- Set a Hold Time, Gate Seal Experiment
- Set a Pack Pressure, Optimize Cpk

Check
- Listen to the Process
- Collect Data
- Chart Trends

Act
- Optimize Cycles
- Troubleshoot Problems
- Improve the Process

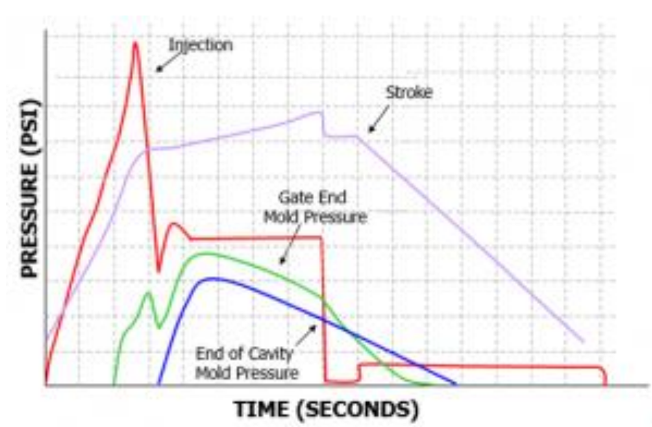
The Power of Decoupled Molding

1. Decoupled molding provides a data-driven method for converting all the variables associated with injection molding to constants.
2. Everyone everywhere develops the same cycle regardless of education, experience, or location.



Once upon a time molders twisted knobs and set timers to make quality parts. The injection molding machine today does essentially what those old machines could do. (Modern machines are admittedly more accurate).

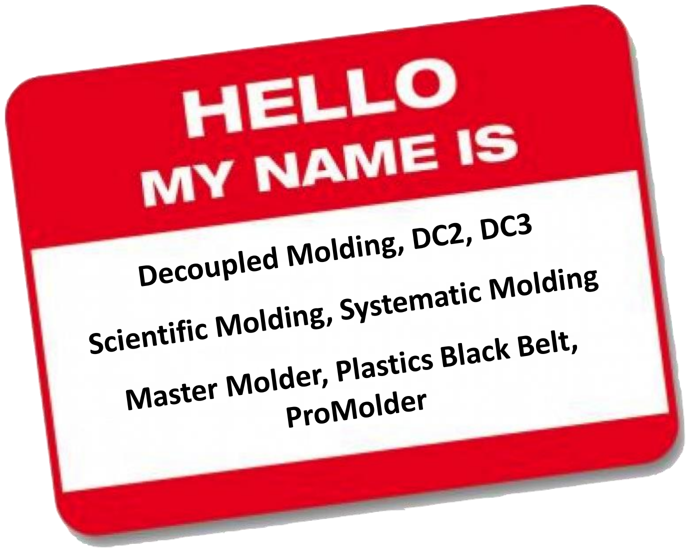
What has changed the way we mold today is the data we can get from the machine and in some cases, the mold.

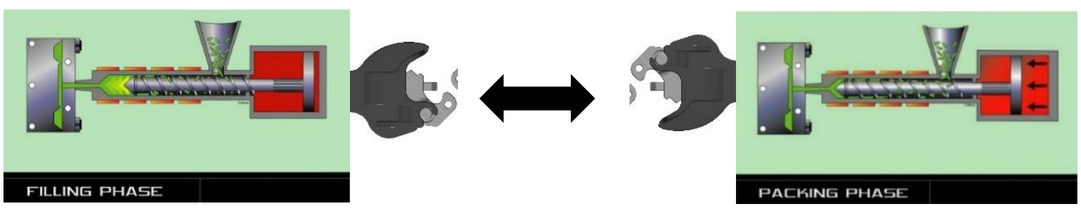


Graphs and data like the ones shown here have been a real game-changer when molding to tight tolerances and specifications.

Decoupled Molding?

How do all the methods for controlling variation differ?





A rose is a rose … The method of separating the fill portion of an Injection Molding Cycle from the Pack and Hold portion of an Injection Molding cycle using position transfer has been marketed under a variety of names. All the names you see below describe this same approach to controlling the variables in an Injection Molding cycle. In this webinar we will use the name Decoupled Molding.

"What's in a **name**? That which we call **a rose**. By **any other name** would smell as sweet." - William Shakespeare, *Romeo and Juliet*

Building an Injection Molding Process, Decoupled Molding

Decoupled, Scientific, and Systematic Molding

Injection Molding process set-up and control has evolved in the past 20 years from black art, based on tribal knowledge, to a science and technology “data-driven” approach. While there are no absolutes in process set-up and control, the industry has adopted an approach known as “decoupled molding” as the method of choice for set-up, validation, and troubleshooting of new and existing injection molding processes. Decoupled molding is also known as scientific or systematic molding.

The Injection Molding process can be described in four simple steps.

1. Melting the raw plastic into a viscous fluid. (A fluid with a thick consistency.)
2. **Injecting the melted plastic into a mold.**
3. Cooling the plastic back into a solid.
4. Ejecting the solid part from the mold.

But, when we looked at the parts of an Injection Molding cycle, the Injection phase (**Injecting the melted plastic into a mold**) of the cycle was broken into three parts.

Injection. (Fill)

Injection. (Transfer)

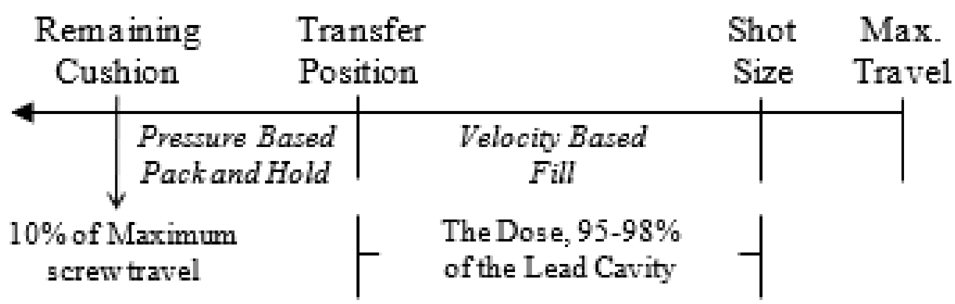
Injection. (Pack and Hold)

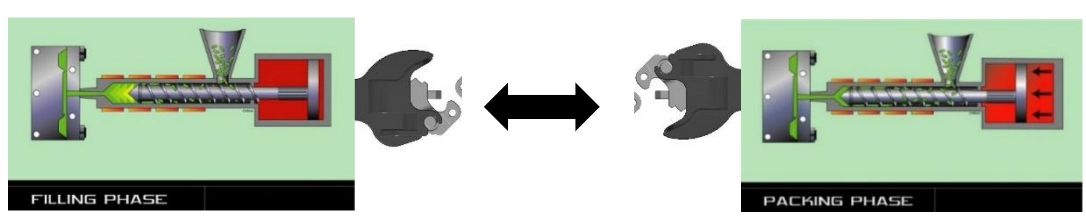
The three “parts” of the Injection phase are needed because during **Injection Fill** (Fill) the machine settings will be focused on the velocity of the screw as it moves forward. A fill pressure limit will be set, but only the needed amount of pressure will be used to keep the screw moving at the desired velocity. The focus on the speed (velocity) of the screw is because this will directly impact several process and material variables and we need only the pressure to fill and empty mold with a thick (viscous) fluid. (Much, much more about all of this later.) The fill portion of Injection is also known as First Stage.

During the **Injection Pack and Hold** portion of the Injection phase, the machine settings are focused on pressure, not velocity. Once the mold fills another 3-5% (103-105% total) of the volume of the molded will be packed into the mold! This requires an amazing amount of pressure but very little screw movement. So, during pack and hold the machine settings are focused to give the same pack pressure and largely ignore screw velocity. The pack and hold portion of Injection is also known as Second Stage.

Transfer is the tool used to determine when to stop using the fill settings and begin using the pack and hold settings. It is critical to get these three Injection phases set correctly so that fill settings are used for fill, and pack and hold settings are used for pack and hold. This might sound simple but MANY molding cycles are not set up this way. We will discuss transfer in detail over the balance of this Chapter.

The figure below illustrates these relationships:



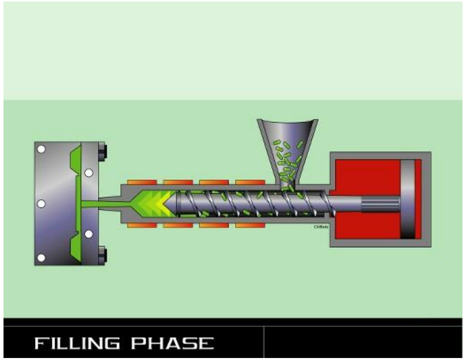


The Parts of a Decoupled Cycle (beginning at fill)

1) Fill (velocity controlled)

Fill aka:
- First Stage
- Injection
- Boost
- Primary

➢ *Fill to Pack transfer (D2 and D3)*

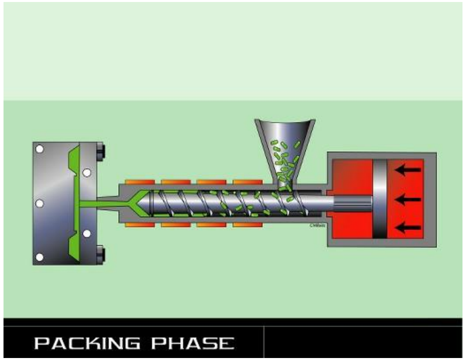


2) Pack and Hold (pressure-controlled)

a) **Pack** (add more material after fill. Why?)

➢ *Pack to Hold transfer (D3 only)*

b) **Hold** the pack (until the gates freeze. Why?)

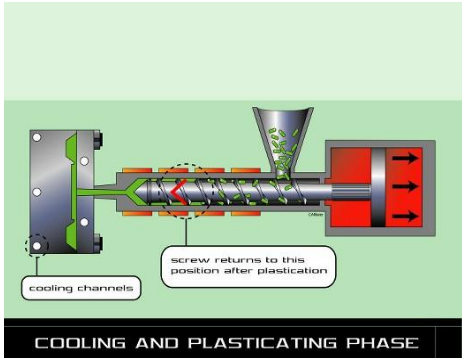


3) Cooling and Plastication

a) the part continues to cool after the gate freezes

b) the injection unit prepares the next shot after the gates freeze

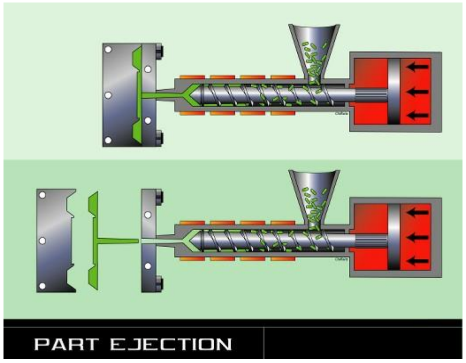
c) the screw pulls back decompressing the melt (suck back)#



4) Eject Part (ASAP)

a) retract injection unit*
b) open the mold
c) eject the part
d) close the mold
e) move injection unit forward*

* if sprue break is selected
\# if suck back is selected



More about the Filling Phase

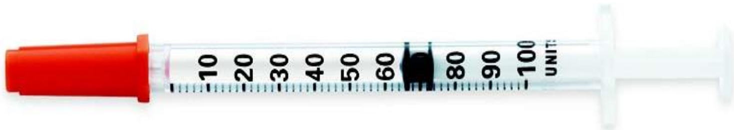
Fill aka:
  First Stage
  Injection
  Boost
  Primary

The Fill Dose

The Dose = The Fill Volume = Shot Size to Transfer Position

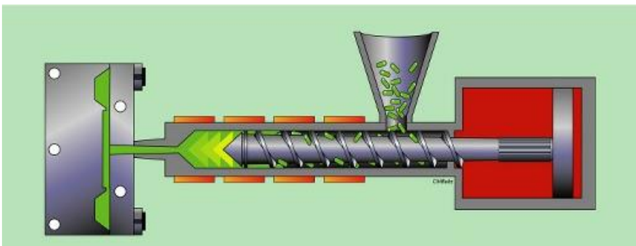
Injecting plastic into a mold during the fill portion of a molding cycle is often referred to as “dosing” the mold. Using this terminology, we will define the correct volume of material added to a mold during filling as “the dose”.

The doctor in the picture at the right is filling a syringe with a dose of medicine. To get the correct amount of medicine the doctor will load the syringe back to some set point. He will then inject the patient until the syringe reaches some desired forward point on the syringe.

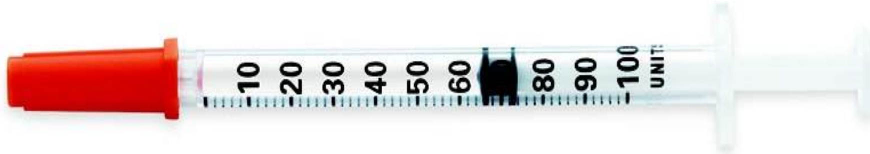


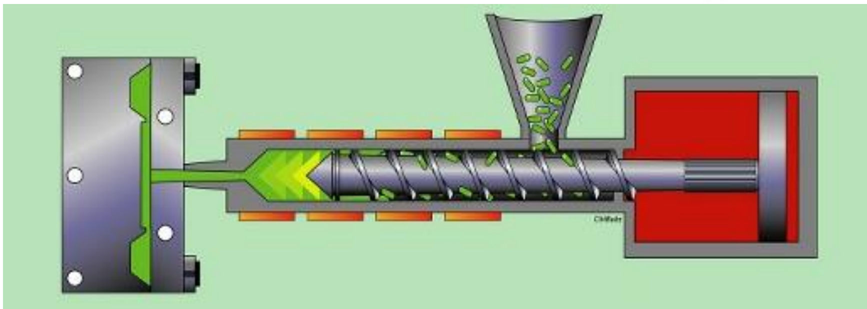
“The dose” in Injection Molding is established the same way. We load the shot by rotating the screw back to “shot size” and then injecting the mold with plastic until we reach the transfer position. The difference between these two positions determines the volume of material injected into the mold during filling, the dose. During a decoupled set up process, we will seek to make an 80% part. We will also make a 95-98% part later in the decoupled set up process. In both cases, we will provide a dose equal to the 80% part and later to the 95-98% part.

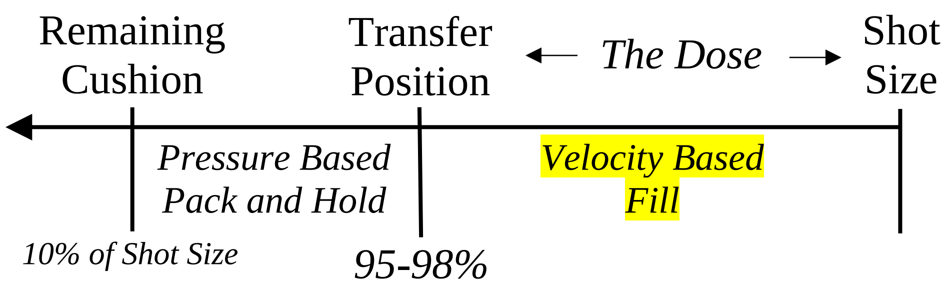
We can adjust the dose by moving either shot size or transfer position.



The Dose Analogy







Why 95-98%?

When we fill, we need to think about when to switch (transfer) from velocity-based fill to pressure-based pack and hold. Just like the warning track in baseball we choose to transfer before we hit the wall (100% full) at full speed. We will choose to slow down at 95 to 98% of the dose.

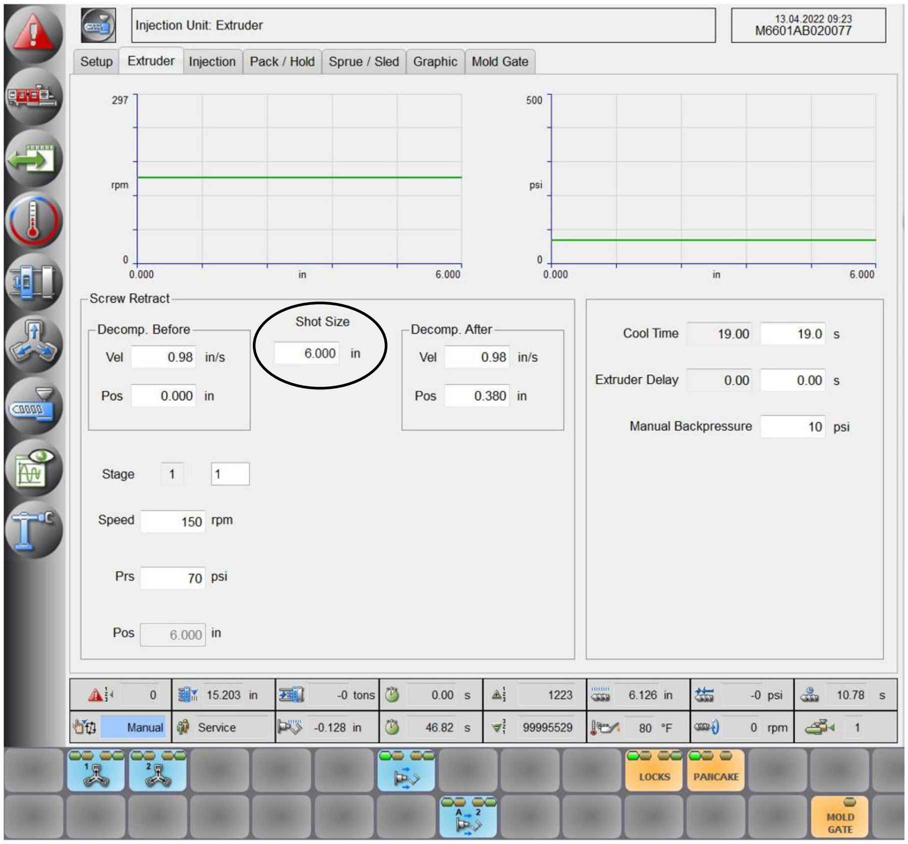
Another example is coming to a stop at a stop sign. It is a good idea to apply the brakes before arriving at the stop sign! Braking early allows for deceleration. Not braking early puts your car in the middle of the intersection!!

https://www.youtube.com/watch?v=EqMXTM2sUrE

Mosaic+ Dose (Shot Size to Transfer, Milacron Fan)

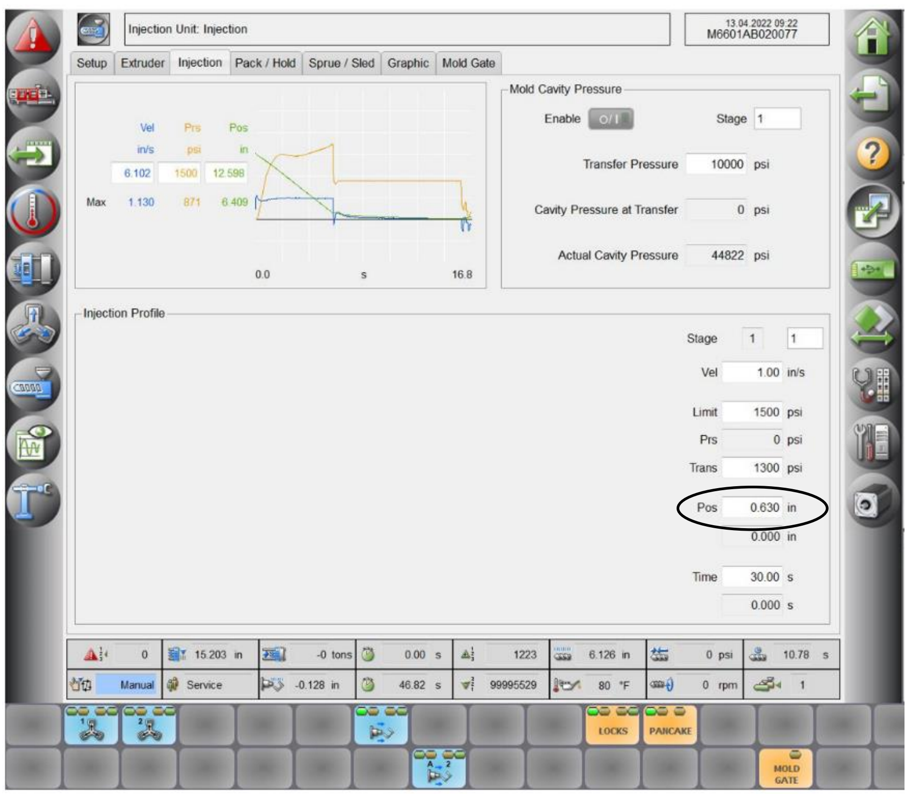
Shot Size

6.000 in.



Position Transfer

0.630 in.



Resulting Fill Dose

5.370 in.

“The Universal Dose”

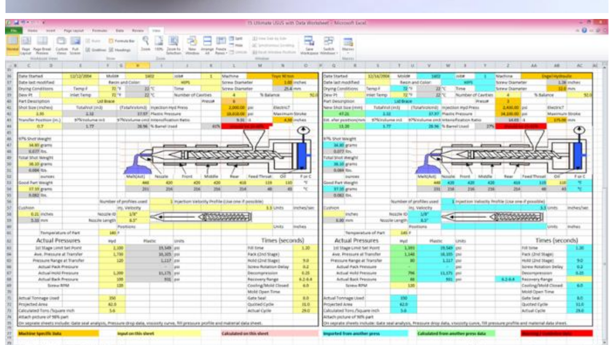
								$V = \pi r^2 h$
Plastics Black Belt		Shot Size	Transfer Position	Fill Dose "h"	Screw Diameter	Screw Diameter	Screw Radius "r"	Shot Volume (V)
		(in)	(in)	(in)	(mm)	(in)	(in)	(in^3)
Machine 1	Robo150a	1.400	0.400	1.000	44	1.73	0.87	2.36
Machine 2	Robo150b	1.700	0.880	0.820	48	1.89	0.94	2.30
Machine 3	Robo110	2.670	0.770	1.900	32	1.26	0.63	2.37
					Difference			0.07
	$h = V / \pi r^2$				% Difference			2.93%

Convert from: Machine 1 (Roboshot 150) (V = π r2 h)							Predict to: Machine 2 (Roboshot 110) (h = V / π r2)							
Shot Size	Transfer Position	Fill Dose 1	Screw 1 Diameter	Screw Diameter	Screw Radius "r"	Shot Volume (V)	Screw 2 Diameter	r	r	r^2	π	Dose 2 h	Transfer Position	Shot Size
(in)	(in)	(in)	(mm)	(in)	(in)	(in^3)	(mm)	(mm)	(in)	(in^2)		(in)	(in)	(in)
1.400	0.400	1.000	44	1.73	0.87	2.36	32	16	0.63	0.40	3.14	1.890	0.350	2.240

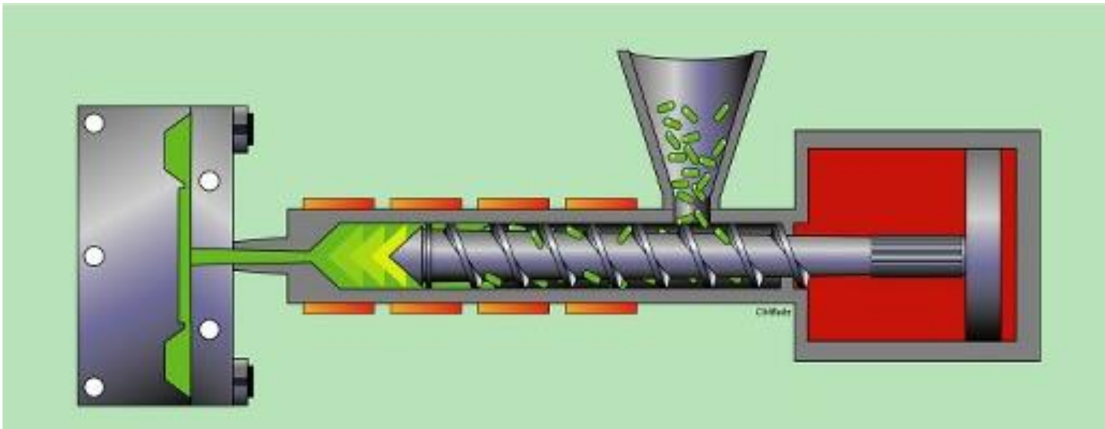
Convert from: Machine 1 (Roboshot 150a) (V = π r2 h)							Predict to: Machine 2 (Roboshot 150b) (h = V / π r2)						Fill	
Shot Size	Transfer Position	Fill Dose 1	Screw 1 Diameter	Screw Diameter	Screw Radius "r"	Shot Volume (V)	Screw 2 Diameter	r	r	r^2	π	Dose 2 h	Transfer Position	Shot Size
(in)	(in)	(in)	(mm)	(in)	(in)	(in^3)	(mm)	(mm)	(in)	(in^2)		(in)	(in)	(in)
1.400	0.400	1.000	44	1.73	0.87	2.36	48	24	0.94	0.89	3.14	0.840	0.350	1.190

Which can holds more?

Universal Set Up



Terminology - % Barrel Usage / Heat History / Residence Time



<u>Barrel Usage</u>

% Barrel Used = (Shot Size-Cushion) (in.) / Maximum Shot Capacity (in.)) / 100

To produce proper mixing and shear on the screw 25% - 65% of the screw should be turned over on every shot.

<u>Heat History</u>

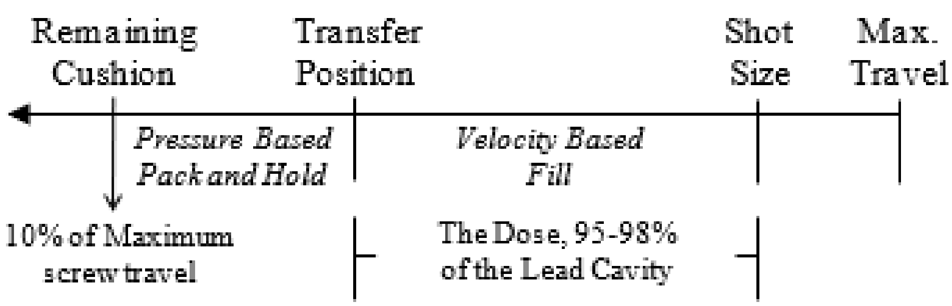
Shots on the Screw = (Maximum Shot Capacity (in.) / ((Shot Size-Cushion) (in.))

The more shots that sit on the screw the more thermal degradation of the plastic.

<u>Residence Time</u>

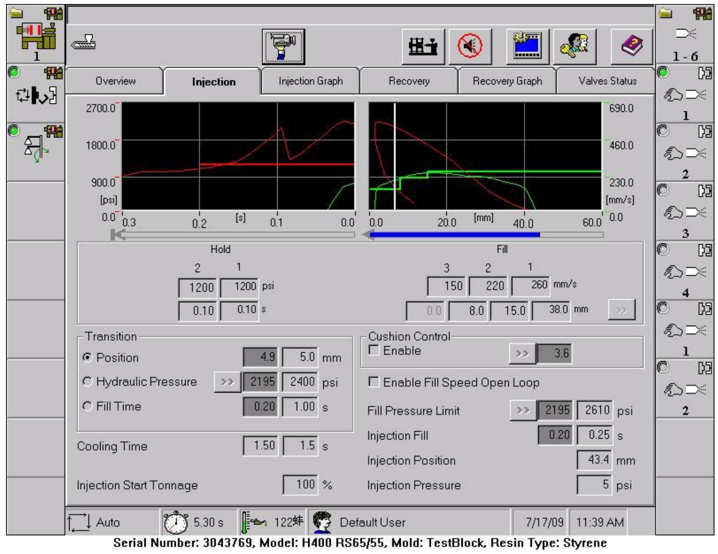
Screw Residence Time = Shots on the Screw x Cycle Time

The amount of thermal degradation will also be impacted by cycle time.



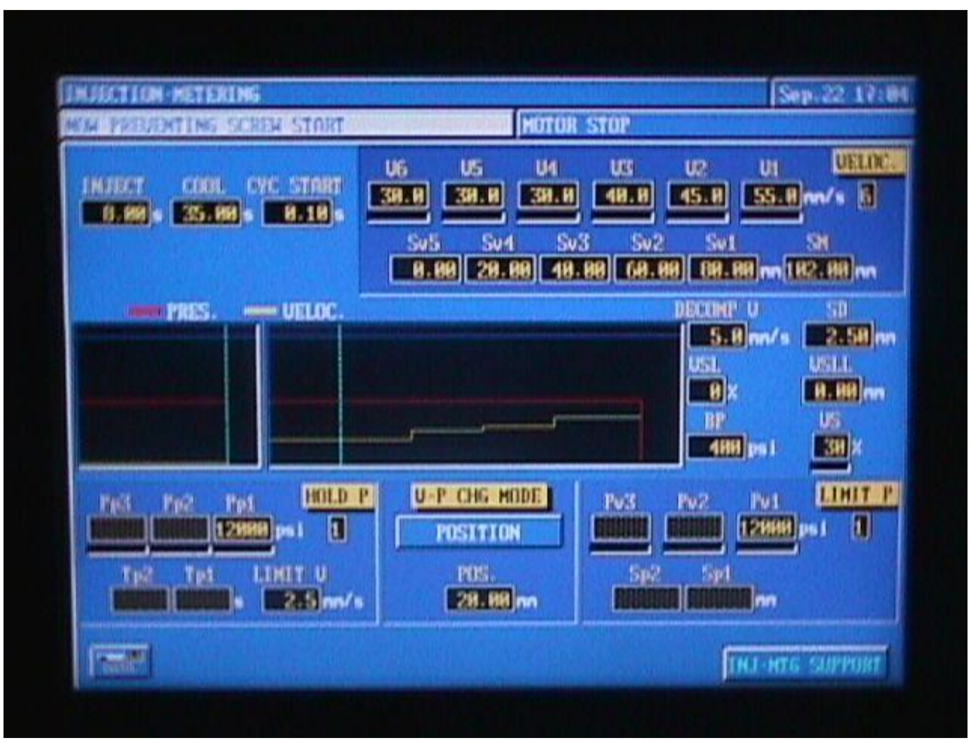
More about Transfer

A key element in decoupled molding is transfer. Transfer is also known as V-P change mode, transition, cutoff, and switchover, to name a few. It seems that every machine manufacturer is compelled to name transfer something cute and unique. Industry process engineers and technicians call transfer, "transfer" almost without exception, go figure.



The Injection Molder screen at the right illustrates the use of transfer. On this machine, transfer is called transition. There are three transfer options on this machine, position, hydraulic pressure, and fill time. Position transfer is currently selected. The transfer option selected is dependent on the part being molded and the cycle being developed.

The Injection Molder screen below also illustrates the use of transfer. On this machine, transfer is called V-P Change Mode. Position transfer is currently selected. A transfer position of 20.00 mm is also specified.



Types of Transfer

There are several ways to determine when the filling portion of the molding cycle is complete, and the pack portion should begin. The following types of transfer are commonly found in use in industry.

Time transfer

- Use on timer-based, solid-state machines.
- Available on many computer-controlled machines.
- Least reliable method of transfer.

Position transfer (used for D2 and D3 cycle development)

- Injects the volume of material between the shot size and the transfer position.
- Relies on the actual cushion for a cycle equaling the transfer position.
- Many manufactures preempt the transfer position and transfer with time. Are you reaching the transfer position?
- Transfer and cushion should never be “0”. The screw must always be applying pressure on the melt until the gates freeze.

Hydraulic Pressure transfer

- Uses hydraulic pressure to gage cavity pressure and shot weight.
- Changes in material viscosity change the relationship between hydraulic pressure and cavity pressure.

Cavity Pressure transfer (used in D3 cycle development)

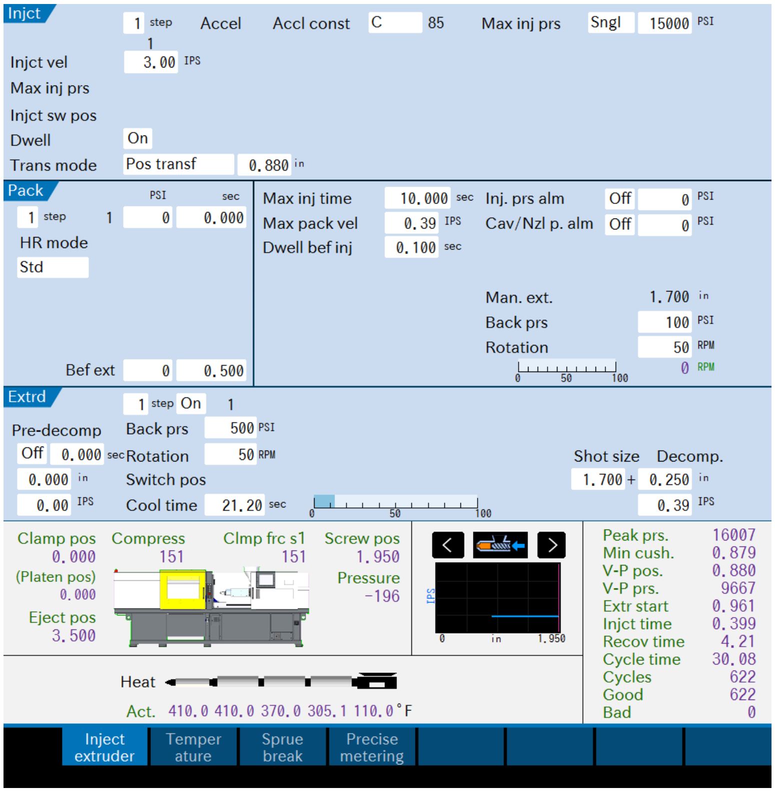
- Best predictor of cavity pressure and cavity pressure consistency.
- To use, molds must be equipped with cavity pressure sensing. (RARE)

External Signal transfer

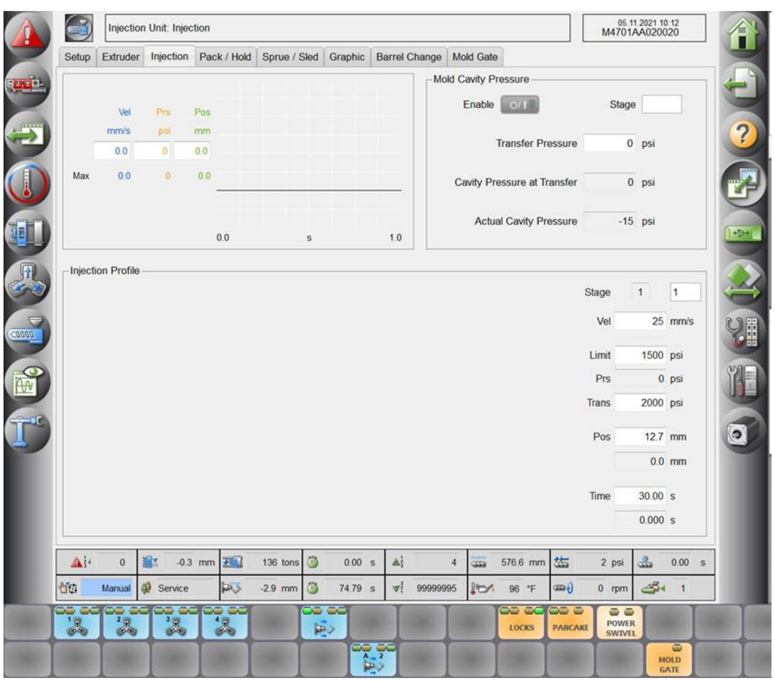
- Best predictor of cavity pressure and cavity pressure consistency.
- To use, molds must be equipped with cavity pressure sensing. (RARE)

Milacron Transfer

Roboshot



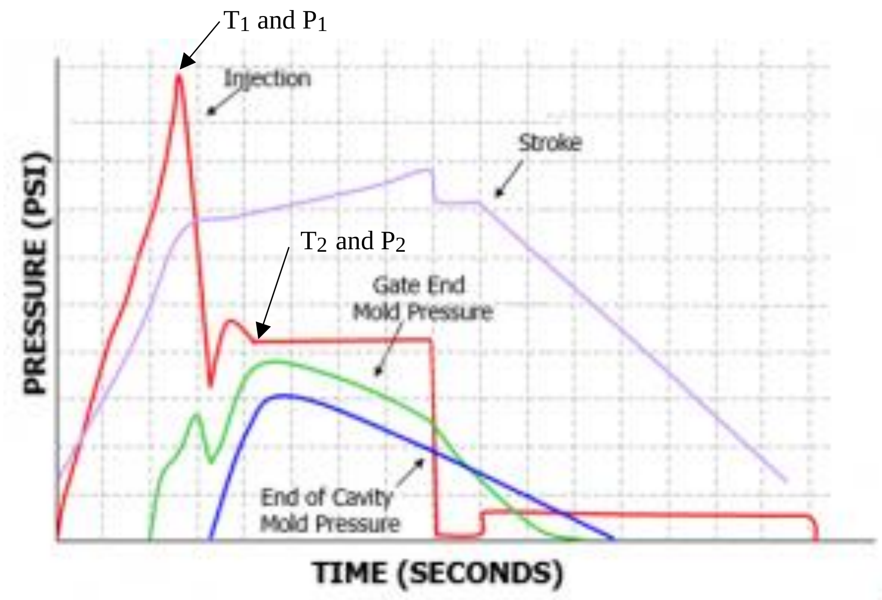
Mosaic+ (Endura Touch)



The IQ Experiments - Switchover Response

Switchover Response Time provides a measure of the performance of the Injection Molding machine at transfer. Lag at transfer (as shown below) will cause undesirable flow hesitation. Slow response time contributes to inconsistent cavity fill and ultimate cavity pressures. Ideally, the Switchover Response will be quick with no lag.

The equation and tables below show data for the calculation of response time in a Hydraulic and Electric machine. The data only differs by the machine intensification ratio.



$$\text{Response Time} = T_2 - T_1 \,/\, P_1 - P_2$$

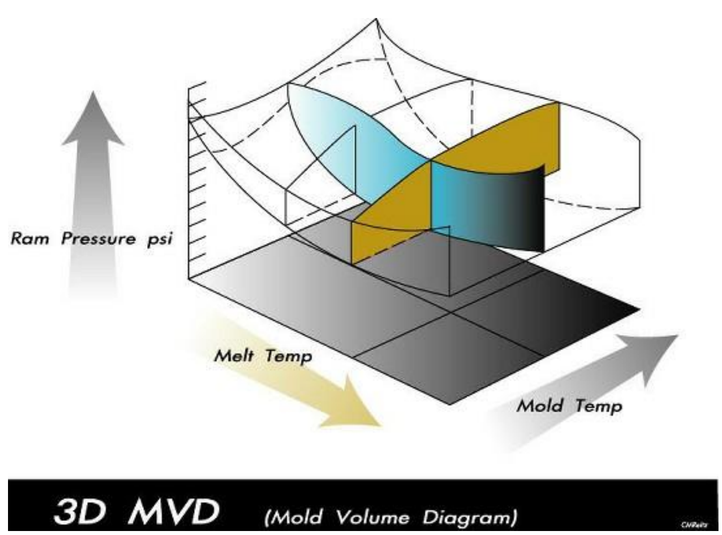
Hydraulic Pressure			Cycle 1	Cycle 2	Cycle 3	Cycle 4	Cycle 5	Average	Minimum	Maximum	Variance
T1	Time at Transfer	sec	0.90	0.89	0.90	0.91	0.89	0.90	0.89	0.91	2.23%
T2	Time after Transfer	sec	1.03	1.02	1.04	1.02	1.01	1.02	1.01	1.04	2.93%
P1	Pressure at Transfer	psi	1,500	1,510	1,500	1,499	1,500	1,502	1,499	1,510	0.73%
P2	Pressure after Transfer	psi	690	700	710	700	700	700	690	710	2.86%
	Switchover Response		0.160	0.160	0.177	0.138	0.150	0.157	0.138	0.177	

Plastic Pressure			Cycle 1	Cycle 2	Cycle 3	Cycle 4	Cycle 5	Average	Minimum	Maximum	Variance
T1	Time at Transfer	sec	0.90	0.89	0.90	0.91	0.89	0.90	0.89	0.91	2.23%
T2	Time after Transfer	sec	1.03	1.02	1.04	1.02	1.01	1.02	1.01	1.04	2.93%
P1	Pressure at Transfer	psi	15,000	15,100	15,000	14,990	15,000	15,018	14,990	15,100	0.73%
P2	Pressure after Transfer	psi	6,900	7,000	7,100	7,000	7,000	7,000	6,900	7,100	2.86%
	Switchover Response		0.160	0.160	0.177	0.138	0.150	0.157	0.138	0.177	

Cycle Development; Decoupled Molding Procedure

Building a Data Driven Decoupled 2 Molding Cycle

1. Bring the machine to steady-state operating conditions.
2. Prepare to capture fill time and pressure at transfer data.
3. Set the machine to transfer on position.
4. Decouple the Cycle by setting "Pack and Hold" pressures to 0.
5. Adjust the Dose to achieve a 95% short lead part.
6. Increase the Fill Pressure to the machine maximum.
7. Optimize Injection Velocities (Injection Velocity Optimization).
8. Optimize the Fill Pressure setting and calculate ΔP.
9. Adjust the Dose to make a 95 to 98% short shot.
10. Add 2nd stage pressure set at ½ Pressure at Transfer.
11. Set hold time by performing a gate seal study.
12. Optimize Dimensions by adjusting Pack Pressure.
13. Determine the Balance of Fill for Multicavity molds.
14. Adjust Screw Speed, Back Pressure, and Barrel Temperatures.
15. Adjust Cooling in the Mold by setting Thermolator temperatures and flow controls.
16. Set Cooling Time and Te to optimize the process window and reduce cycle time.
17. Do cycle quick checks to assure all settings are optimized.



Translating the Controller

Each Injection Molding machine does the same basic functions as we discussed previously. The actual control settings and computer logic used to control these basic functions vary widely among machine manufacturers. For every Injection Molding machine, it is necessary to "translate" the settings on a machine to the standard terminology we have just discussed. The following is a list of key settings we will need to know to apply the theory of Decoupled molding any machine. It would be great if Google Translate work for Injection Molding machines!

Machine Settings – Settings we will need to know

Determine how to:

- Navigate within the Control Software.
- Enter Data (make changes) in data fields.

Determine the location of the following machine control settings:

- Injection Pressure(s)
- Injection Speed(s) (Injection Velocities)
- Transfer program (ie: time, position, pressure, etc.)
- Transfer Position
- Pack Pressure
- Hold Time

Also find:

- Cooling Time
- Shot Size

Determine how to monitor:

- Actual Fill Time
- Actual Pressure at Transfer
- The Cushion

Determine the Machine Intensification Ratio

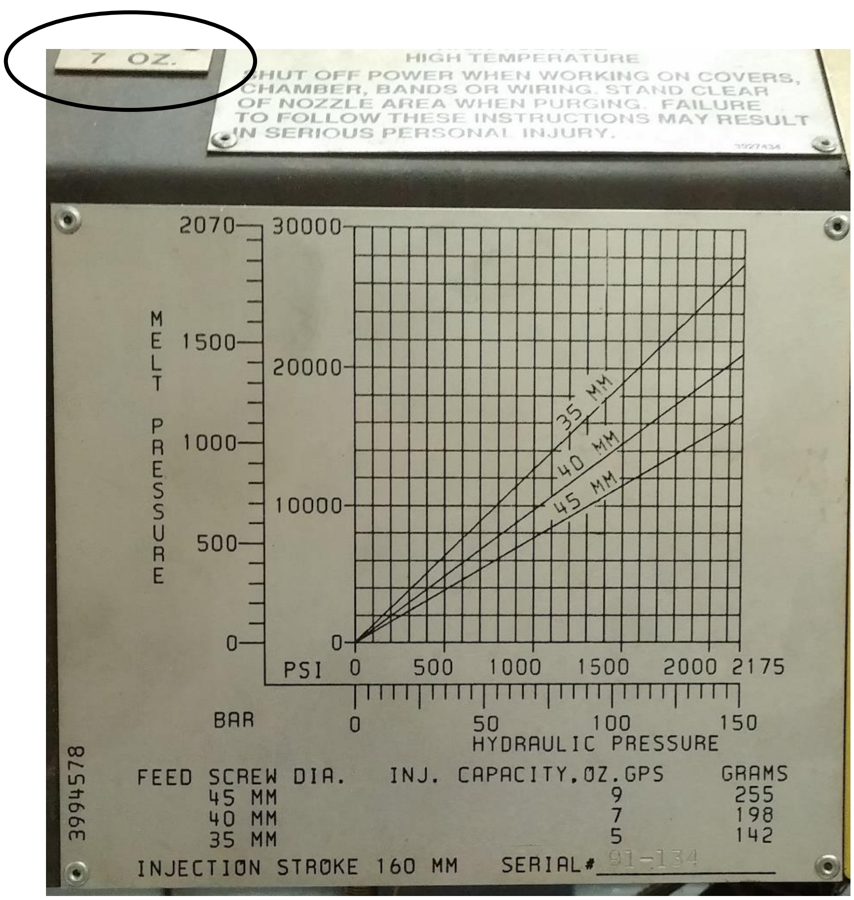
How to Calculate Machine Intensification Ratio

The ratio, P_{melt} /$P_{hyd.}$ is referred to as the Intensification Ratio of a machine.

The graph below shows the relationship between melt pressure and hydraulic pressure for three Injection Unit options on an Injection Molding machine. The three injection units have a 35mm, 40mm, or 45mm screw. (It is common practice for molding machine manufacturers to provide injection unit multiple options for each clamp size.)
In this case this machine has the 40mm screw. The small tab at the extreme top of the picture reads "7 oz." The table at the bottom of the picture ties the 40mm screw to the 7oz. injection capacity.

The 40mm line intersects the hydraulic pressure on the x axis at 2,175 psi and the melt pressure on the y axis at 21,750 psi.

Intensification Ratio = P_{melt} /P_h =21,750 PSI / 2,175 PSI = a 10:1 ratio.



Determine the Dose - Short Shot Methods

Determine the Dose by **one** of the following 4 methods:

1. **Air Shot Weight**
2. Portability from another machine
3. Shot Volume Calculation
4. Estimated Fill Time

Method 1 - Air Shot Weight

1. Determine the weight of a shot (part, runner, sprue)
2. Make a shot. Record the Shot Size position.
3. Purge the shot fully onto the bed of the machine. Assure the cushion is zero.
4. Weigh the purge.
5. Repeat the procedure, adjusting shot size until the shot weight equals the total weight of the shot. The screw travel will be your full Dose estimate.
6. Set up a decoupled test run with Shot Size to Transfer (the Dose) set to 95% of the full dose you determined. Be sure to leave some cushion.
7. Attempt single cycles. If the total shot ejects cleanly, repeat single cycles until the mold is warmed by the melt. (Approximately 10 cycles).
8. Adjust transfer as necessary to achieve an 80% full part.

Determine the Dose - Short Shot Methods

Determine the Dose by **one** of the following 4 methods:

1. Air Shot Weight
2. **Portability from another machine**
3. Shot Volume Calculation
4. Estimated Fill Time

Method 2 - Portability from another machine

								$V = \pi r^2 h$
Plastics Black Belt		Shot Size	Transfer Position	Fill Dose "h"	Screw Diameter	Screw Diameter	Screw Radius "r"	Shot Volume (V)
		(in)	(in)	(in)	(mm)	(in)	(in)	(in^3)
Machine 1	Robo150a	1.400	0.400	1.000	44	1.73	0.87	2.36
Machine 2	Robo150b	1.700	0.880	0.820	48	1.89	0.94	2.30
Machine 3	Robo110	2.670	0.770	1.900	32	1.26	0.63	2.37
					Difference			0.07
	$h = V / \pi r^2$				% Difference			2.93%

Determine the Dose - Short Shot Methods

Determine the Dose by **one** of the following 4 methods:

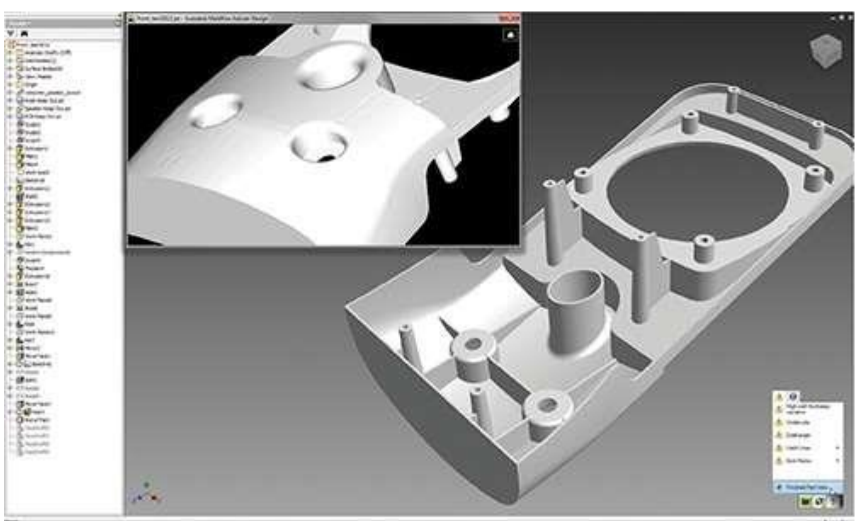
1. Air Shot Weight
2. Portability from another machine
3. **Shot Volume Calculation**
4. Estimated Fill Time

Method 3 - Shot Volume Calculation

1. Determine the weight of a shot (part, runner, sprue)
2. Determine material density (melt density is more accurate, if available).
3. Calculate shot volume needed (Volume, V = Shot Wt. / Density)
4. Determine the screw diameter, calculate radius (r = Ø/2)
5. Calculate the cross-sectional area of the injection unit barrel ($A = \pi r^2$)
6. Calculate the shot stroke needed to provide the shot volume previously calculated.
   (if Volume, V = Ah, then h = V/A, where h is the shot stroke)
7. Determine the initial transfer point by subtracting shot stroke from $^2/_3$ maximum shot capacity.
8. Attempt a single cycle; adjust transfer as necessary to achieve an 95% full part.

Caution: be sure to use like units of measure (inch, cm, mm) or to convert if necessary.

NOTE: This method is most commonly used when CAD estimates are available.



Determine the Dose - Short Shot Methods

Determine the Dose by **<u>one</u>** of the following 4 methods:

1. Air Shot Weight
2. Portability from another machine
3. Shot Volume Calculation
4. **Estimated Fill Time**

Method 4 - Estimated Fill Time

1. Estimate fill time based on the size of the part.

   Tiny = 0.5 sec = one Mi….

   Small = 1 sec = one Mississippi

   Medium = 2 sec = one Mississippi, two Mississippi

   Large = 3 sec = one Mississippi, two Mississippi, three Mississippi

   Huge = 5 sec = one Mississippi, two Mississippi … to five Mississippi
2. Make a shot at $^2/_3$ maximum shot capacity.
3. Manually inject the shot for the fill time estimated above, record the cushion.
4. If the part is <u>just short</u>, use the observed cushion as your transfer position. If the part is <u>full or flashed</u>, reduce the cushion by 10% of the shot size and enter this as the initial transfer position. If the part is <u>short by less than half of the part</u>, add 10% to the observed cushion and enter this as the initial transfer position.
5. Attempt a single cycle; adjust transfer as necessary to achieve an 95% full part.

*NOTE: **<u>IF THE PARTS STICK</u>**. The 95% part does not need to be exactly 95%. If producing an 95% part makes the part stick or otherwise keeps the part from ejecting, continue to adjust the dose size until the machine will cycle.*

*NOTE: **<u>THE CUSHION</u>**. At this point don't become concerned with the cushion. Once we have all the fill parameters set, we will adjust the final cushion. <u>Just be sure the cushion is not zero.</u>*

*NOTE: **<u>EXACTLY 95%?</u>** No, you don't need to weight the part. 95% is a visual estimate. The 95% part is used as a starting point for the next experiments. The 95% part should be a part that is not full but mostly full. It should be a part that ejects well if possible.*

Milacron

Processing 1

Control Utilization and Cycle Development

Day 2

Cycle Development

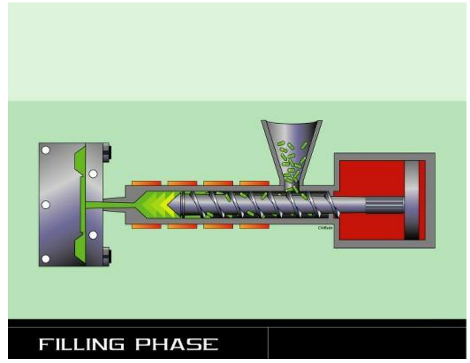
Wednesday Cycle Development

7:30 a.m.	Check in with Refreshments
8:00 a.m.	- Quick Review with Q & A
	- Is a Cycle Pressure Limited?
	Preview the Pressure Limited by fill time experiment
	Lab Session 3: Pressure Limited by fill time analysis
	Review Data from Pressure Limited by fill time
	- Delta P
	- Fill Pressure "On the Roof"
9:45 a.m.	**Break/Networking**
10:00 a.m.	- Using Graphs to Analyze Fill
	Preview Injection Pressure graph analysis
	Lab Session 3: Pressure Limited by IP graph analysis
	Review Data from Injection Pressure graph analysis
	- The Fill (95-98%) Part
	Preview Recoupling the Cycle
	Lab Session 4: Recoupling the Cycle
	Review Data from Recoupling the Cycle
Noon	**Lunch**
12:30 p.m.	- Orientation and Injection Velocity Optimization
	Preview of Injection Velocity Experiment
	Lab Session 5: The Injection Velocity Experiment
	Review Injection Velocity Optimization Data
	- Injection Velocity Profiling
	- What is a short shot series?
2:30 p.m.	**Break/Networking**
2:45 p.m.	Preview The Gate Seal Experiment
	Lab Session 6: Gate Seal Experiment
	Review Gate Seal Data
	- Amorphous and Crystalline Shrinkage
	- PvT Relationships
	Preview Optimizing Pack Pressure
	Lab Session 7: Pack Pressure, Maximizing C_{PK}
	Review Optimizing Pack Pressure
	- End of Day Wrap-Up with Q & A
5:00 p.m.	**Adjourn**

The Parts of a Decoupled Cycle (beginning at fill)

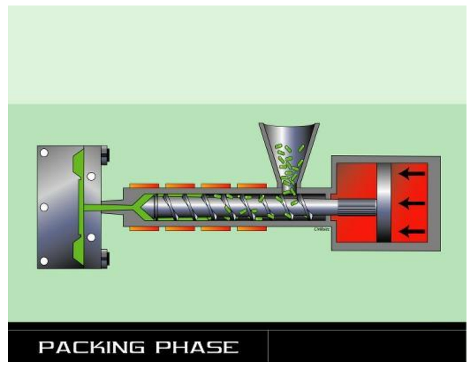
1) Fill (velocity controlled)

Fill aka:
- First Stage
- Injection
- Boost
- Primary



➢ *Fill to Pack transfer (D2 and D3)*

2) Pack and Hold (pressure-controlled)

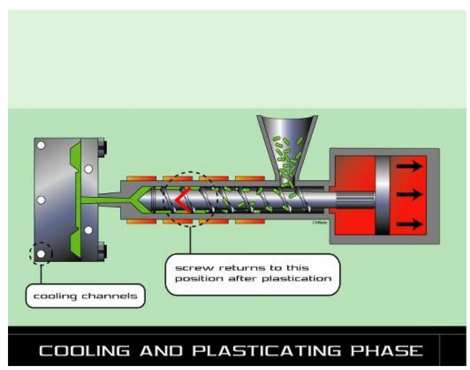


a) **Pack** (add more material after fill. Why?)

➢ *Pack to Hold transfer (D3 only)*

b) **Hold** the pack (until the gates freeze. Why?)

3) Cooling and Plastication

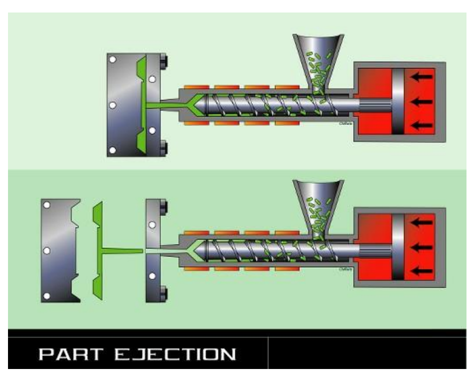


a) the part continues to cool after the gate freezes

b) the injection unit prepares the next shot after the gates freeze

c) the screw pulls back decompressing the melt (suck back)#

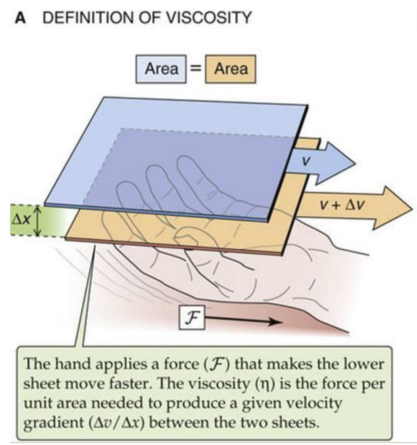
4) Eject Part (ASAP)



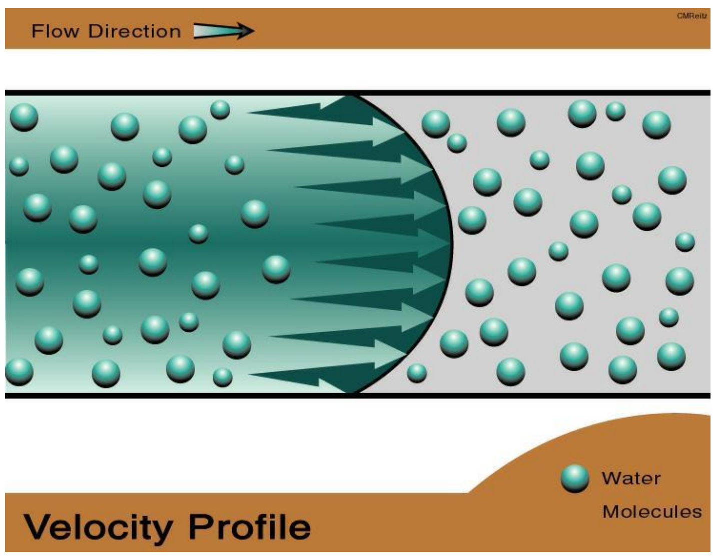
a) retract injection unit*
b) open the mold
c) eject the part
d) close the mold
e) move injection unit forward*

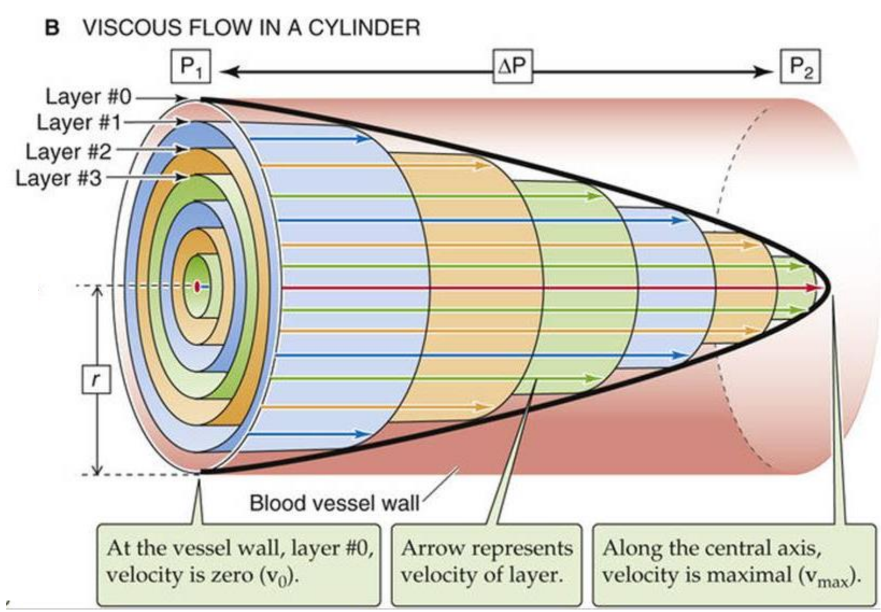
* if sprue break is selected
\# if suck back is selected

Injection Velocity - Laminar Runner Flow

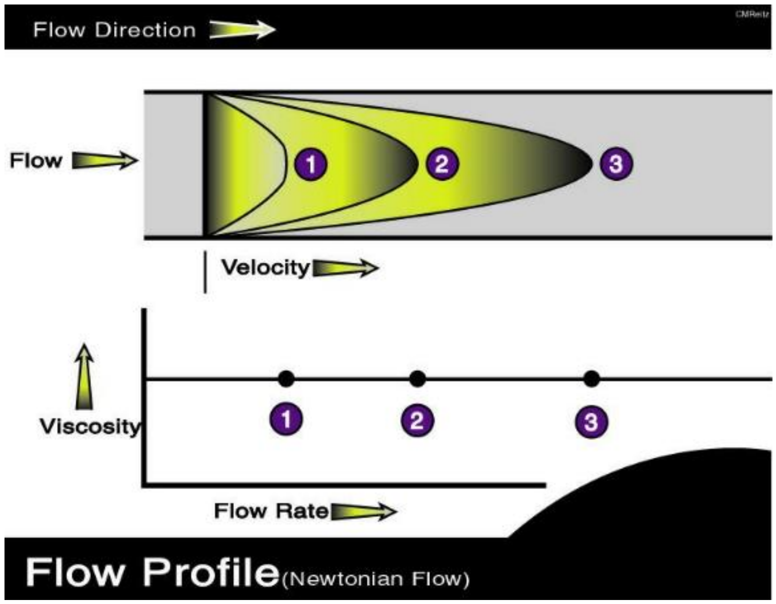


Water in a pipe will flow at different speeds. The water near the wall will be slowed by friction will the wall. This slowing by friction is known as "wall drag".

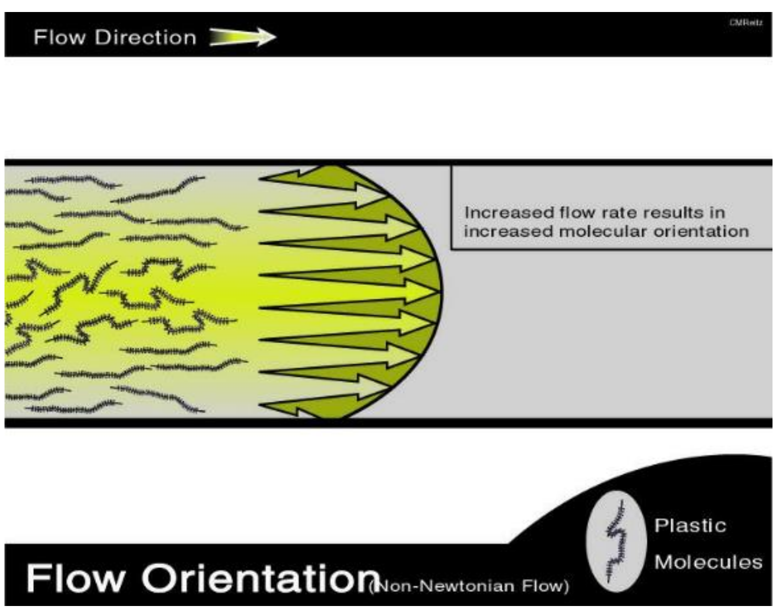




The faster the water flows (increasing velocity) the greater the speed difference between the water near the wall and the water near the center of the channel. This difference in speed is known as “shear”.

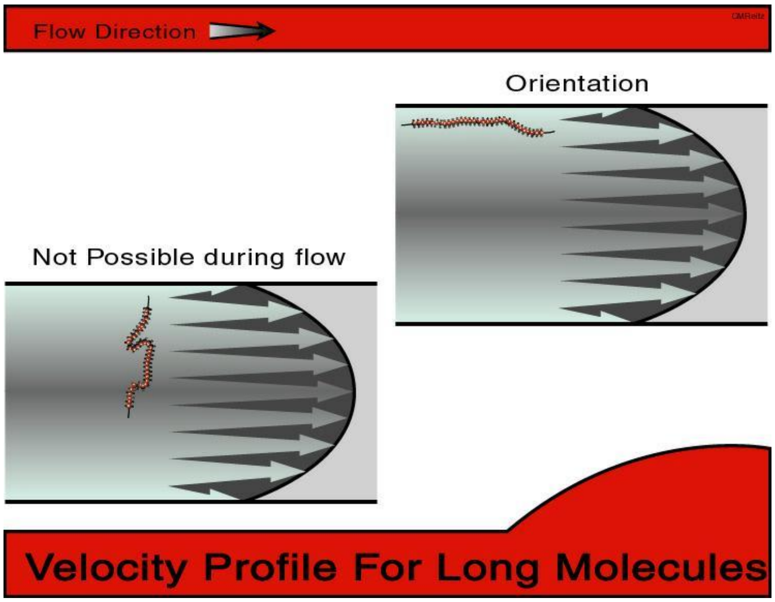


Polymers (plastic) experience the same wall drag in runners, gates, etc. They also exhibit this laminar flow profile.

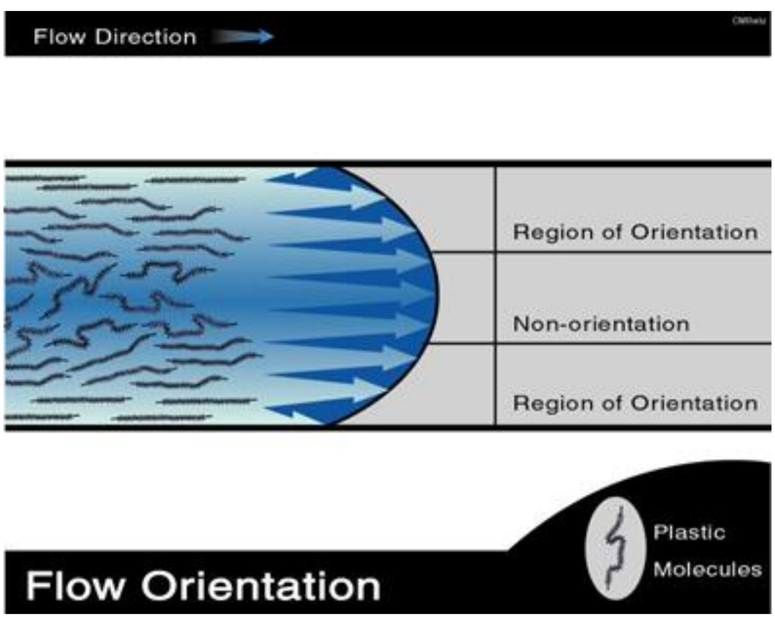


Orientation and Viscosity

When laminar flow occurs, long molecules like polymers (plastics) tend to align in the direction of flow. This is known as Orientation.

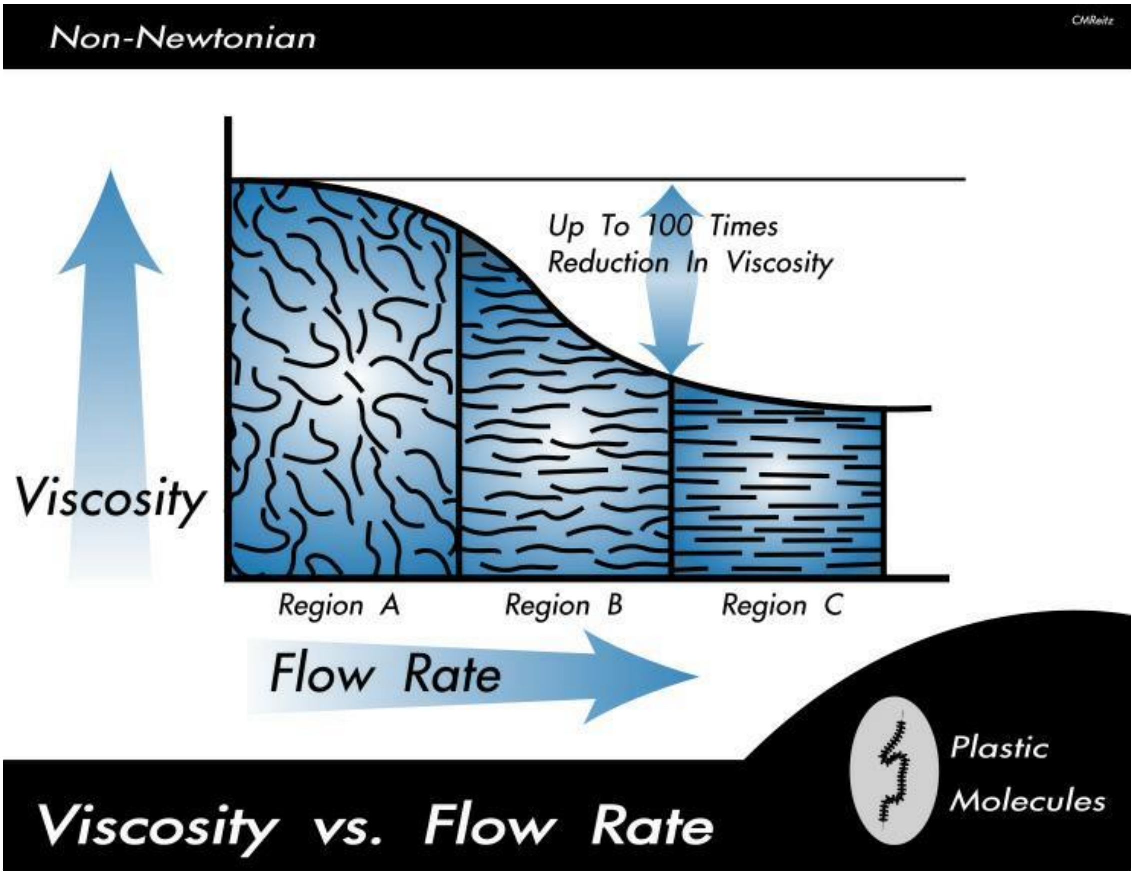


The greater the difference in flow rate (shear) the more oriented the molecules in that region. The molecules in the center of flow are less oriented than the molecules near the wall.



Logs in Boom, Susquehanna River, Williamsport, Pa.

The graph below describes the correlation between flow rate and the degree of orientation. The material viscosity also decreases with increases in flow rate and orientation.



Power Law Coefficient of Commonly Used Polymers

Polymer	Consistency Index	Power Law Coefficient	Temperature Region, C
HDPE	2.0×10^4	0.41	180
LDPE	6.0×10^3	0.39	160
Nylon 66	6.0×10^2	0.66	290
PC	6.0×10^2	0.98	300
PP	7.5×10^3	0.38	200
PS	2.8×10^4	0.28	170
PVC	1.7×10^4	0.26	180

Source: International Plastics Handbook, Table 3.11

Establish the Optimum Injection Velocity

VELOCITY OPTIMIZATION

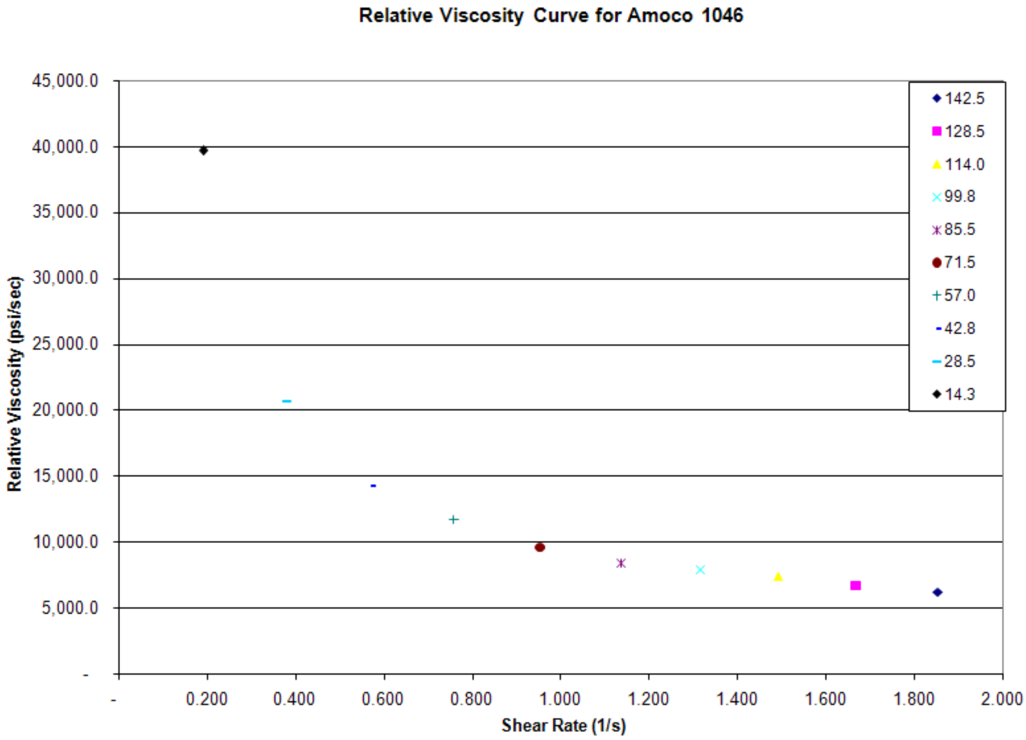
Purpose: To select the optimum injection velocity.

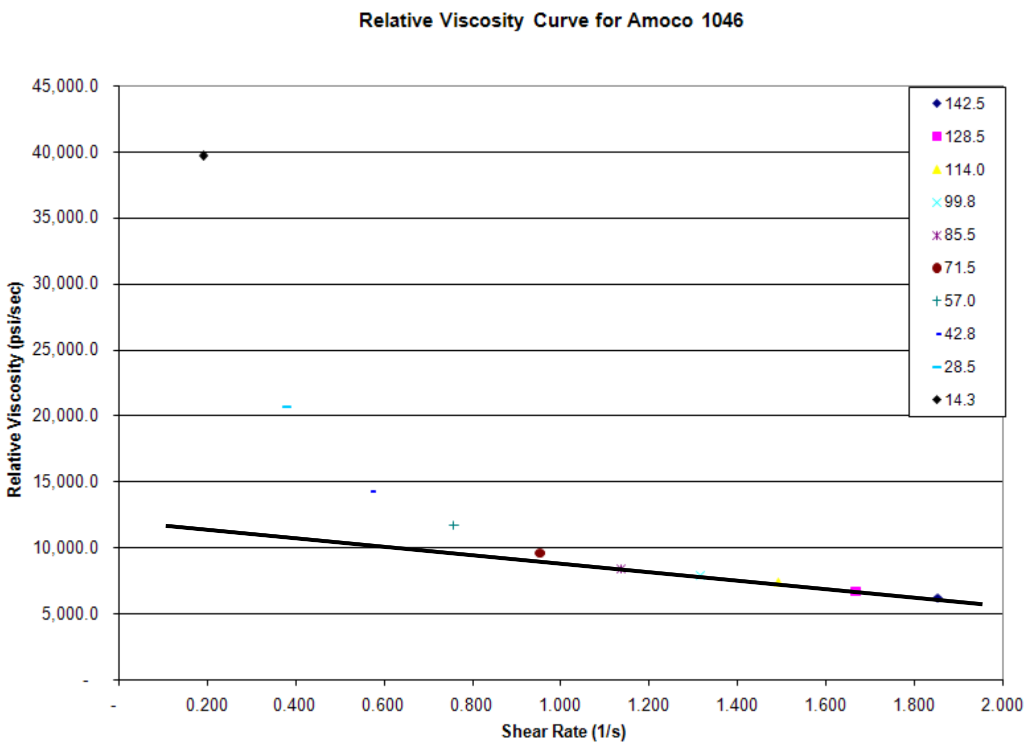
Method: Measure actual melt viscosity using the Injection Molding Machine as a Rheometer.

Procedure:
1. Set Injection Fill to maximum.
2. Determine the maximum Injection Speed for the machine. Divide the maximum speed into ten equal test points. Beginning with the slowest velocity, inject parts using each Injection Velocity.
3. Record Fill Time and Pressure at Transfer for each Injection Velocity.
4. Repeat step 2 until maximum Injection Speed is reached.
5. Calculate viscosity and shear rate.
6. Graph the data.

Maximum Injection Velocity:			150.00	(mm/sec)
90%	Injection Velocity >		135.00	(mm/sec)
Shot Number	Fill Time (sec)	P at Transfer (psi)	Shear Rate (1/sec)	Rel Viscosity (psi*sec)
3	0.660	12,960	1.52	8,554
4	0.660	12,960	1.52	8,554
5	0.660	12,960	1.52	8,554
Avg	0.660	12960.00	1.52	8,554
StDev	0.00	0.00	0.00	-
80%	Injection Velocity >		120.00	(mm/sec)
Shot Number	Fill Time (sec)	P at Transfer (psi)	Shear Rate (1/sec)	Rel Viscosity (psi*sec)
3	0.740	11,480	1.35	8,495
4	0.740	11,480	1.35	8,495
5	0.740	11,480	1.35	8,495
Avg	0.740	11,480	1.35	8,495
StDev	0.00	-	0.00	-

Setting Optimum Injection Speeds, Graphing the Data





Injection Velocity Optimization

Injection Velocity	Shot No.	Fill Time	Ave.	Pres. @ Transfer	Ave.
10%	3				
	4				
	5				
20%	3				
	4				
	5				
30%	3				
	4				
	5				
40%	3				
	4				
	5				
50%	3				
	4				
	5				
60%	3				
	4				
	5				
70%	3				
	4				
	5				
80%	3				
	4				
	5				
90%	3				
	4				
	5				
100%	3				
	4				
	5				

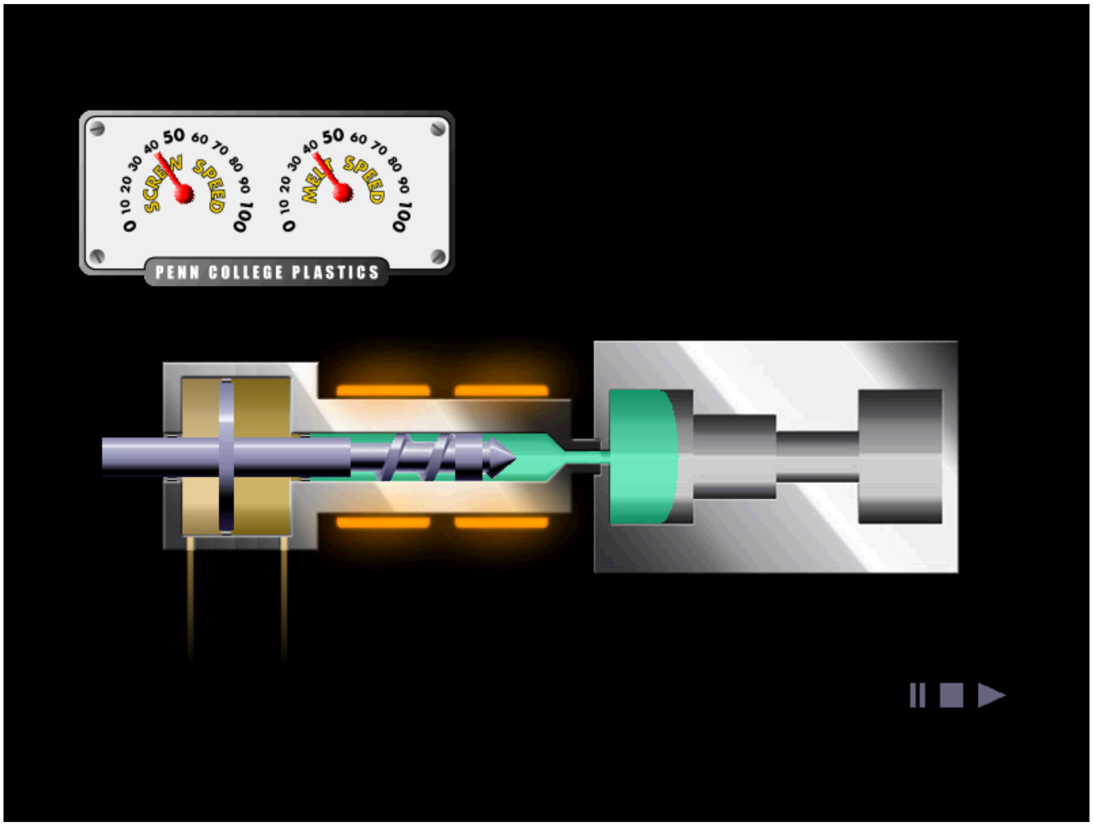
Constant Melt Velocity - Controlling Part Orientation

Injection Velocity Profiling

What is a short shot series?

To achieve **constant melt velocity,** it is necessary to profile the Injection Speeds (Injection Velocities) on an Injection Molding machine.

The result of a constant melt velocity profile is constant orientation throughout the part.



Notes:

The IQ Experiments - Injection Speed Linearity

Injection Velocity Linearity Experiment

1. Set up a decoupled cycle with pack pressure and hold time set to 0.

2. Assure the lead part is roughly 95% full.

3. Set the fill speed to 10% of machine maximum.

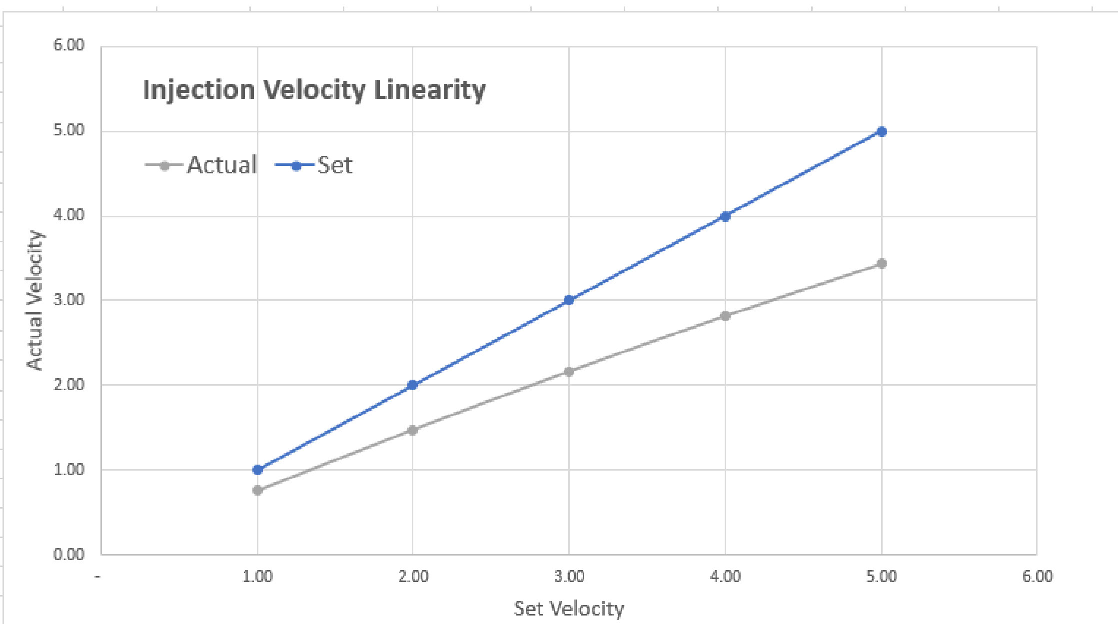
4. For 5 cycles, record the shot size, transfer position, and fill time.

5. Repeat for Injection Speeds 20% through 90% of the machine maximum.

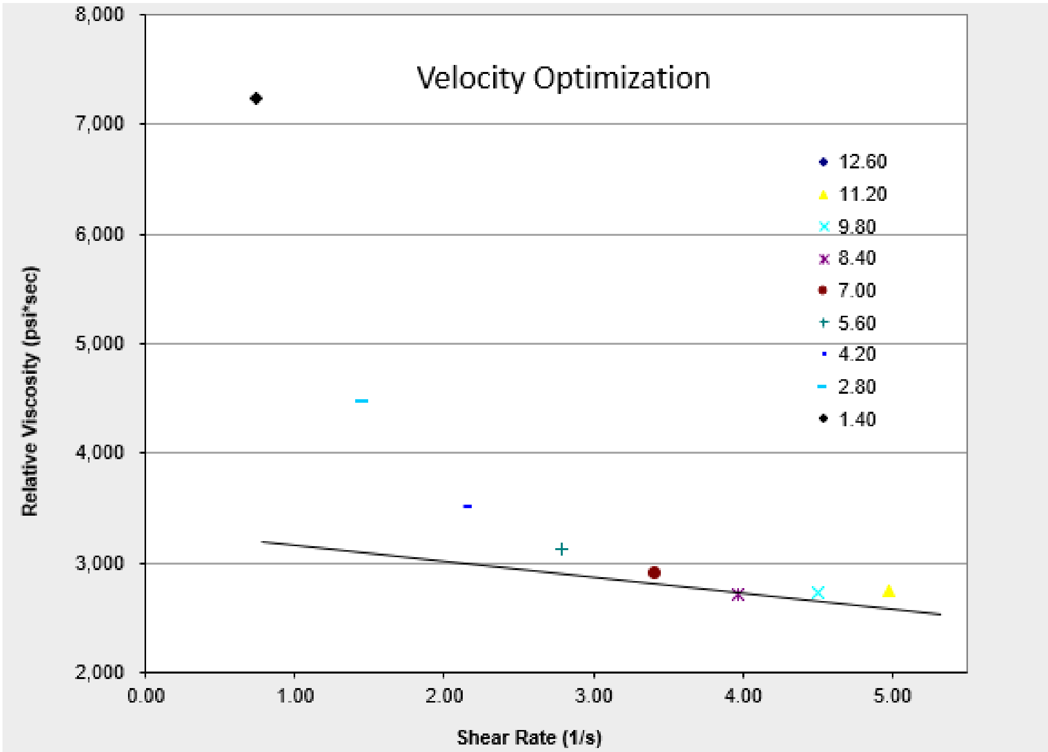
5. Calculate and plot the actual velocity observed vs. the set velocity at each speed.

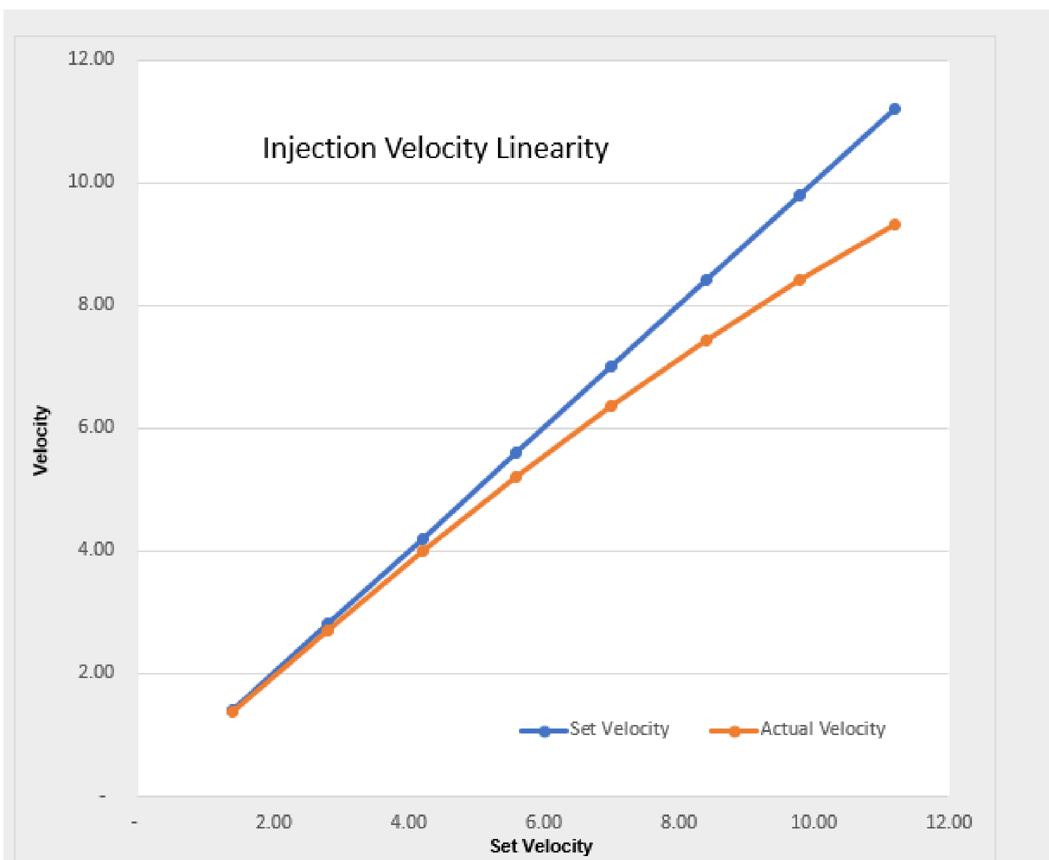
NOTE: The observed variation should be recorded over time for each speed.

Shot No.	Set Velocity in/sec	Shot Size mm	Transfer mm	Dose mm	Fill Time sec	Actual Velocity mm/sec	% of Set %
1	1.00	1.700	0.880	0.820	1.090	0.75	75%
2	2.00	1.700	0.880	0.820	0.556	1.47	74%
3	3.00	1.700	0.880	0.820	0.379	2.16	72%
4	4.00	1.700	0.880	0.820	0.291	2.82	70%
5	5.00	1.700	0.880	0.820	0.239	3.43	69%
		Full Shot >		6.98			
		% Barrel Usage >		11.7%			



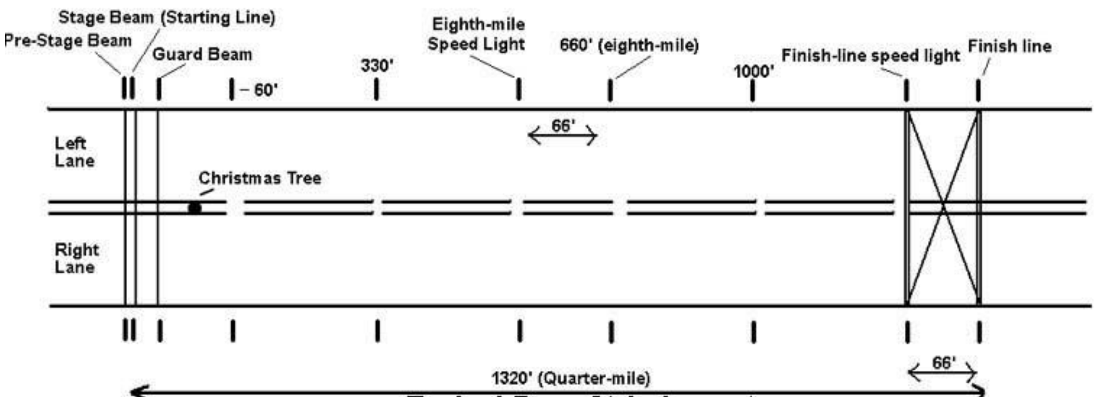
Injection Velocity Optimization (w/ Injection Velocity Linearity)





Fill Time as a measure of Screw Speed (Injection Velocity)

In drag racing, we can calculate the average speed of a car (or motorcycle) by clocking the elapsed time from start to finish of a drag strip of known length.



Typical Drag Strip Layout

For example, look at the two cars shown at the right.

Car 1

3.534 sec = 213.27 mph

Car 2

3.555 sec = 212.43 mph

We use fill time to track the speed of the screw during injection in the same way as the drag strip uses elapsed time to track the speed of a car.

The Pressure Limited Experiment (2 Ways)

Increase the 1st stage Injection Pressure until fill time does not decrease.

Using the following table, set the 1st stage Injection Pressure should be set at 30% of the machine maximum. Increase the 1st stage Injection Pressure in increments of 5% of the machine maximum until fill time does not decrease.

5% =	PSI	30% =	PSI

Sample ID	Cycle	Set Pressure (PSI)	Fill Time (sec)	Transfer Pressure (PSI)
Initial Setting	3			
(30% of max) psi	4			
	5			
(Initial + 5%) psi	3			
	4			
	5			
(Initial + 10%) psi	3			
	4			
	5			
(Initial + 15%) psi	3			
	4			
	5			
(Initial + 20%) psi	3			
	4			
	5			
(Initial + 25%) psi	3			
	4			
	5			
(Initial + 30%) psi	3			
	4			
	5			
(Initial + 35%) psi	3			
	4			
	5			
(Initial + 40%) psi	3			
	4			
	5			
(Initial + 45%) psi	3			
	4			
	5			
(Initial + 50%) psi	3			
	4			
	5			
(Initial + 55%) psi	3			
	4			
	5			
(Initial + 60%) psi	3			
	4			
	5			

(NOTE 1: when fill time no longer is decreasing you should notice a gap of several hundred PSI between the set Injection Pressure value and the hydraulic pressure at transfer.)
(NOTE 2: All parts should still be short.)
(NOTE 3: At no time should the final screw cushion be 0.)

Fill Pressure Delta P

Pressure Limit (Mosaic+ Table Method, Milacron Fan)

Set P	Cycle	Fill Time	Pat Transfer	Delta P
500	1	12.464	474	26
500	2	12.455	475	25
500	3	12.550	474	26
600	1	9.390	573	27
600	2	9.396	576	24
600	3	9.424	573	27
700	1	7.674	673	27
700	2	7.698	671	29
700	3	7.721	671	29
800	1	6.665	769	31
800	2	6.602	702	98
800	3	6.635	768	32
900	1	6.344	868	32
900	2	6.352	868	32
900	3	6.348	867	33
1,000	1	6.324	910	90
1,000	2	6.320	904	96
1,000	3	6.324	913	87
1,100	1	6.320	903	197
1,100	2	6.320	907	193
1,100	3	6.323	914	186

Increase the 1st stage Injection Pressure until fill time does not decrease.

PRESSURE LIMITED: Using the following table, set the 1st stage Injection Pressure should be set at 30% of the machine maximum. Increase the 1st stage Injection Pressure in increments of 5% of the machine maximum until fill time does not decrease.

DELTA P (ΔP): To assure the molding machine has enough reserve to meet the changing needs of material viscosity changes continue to raise pressure until the two following criteria are met.

A) fill time stops dropping and becomes constant; and

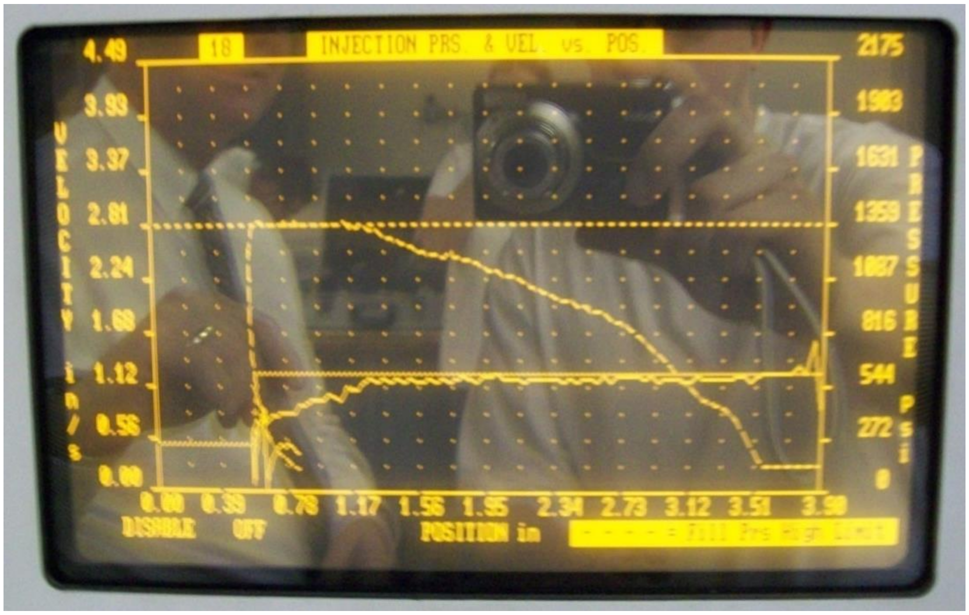
B) peak pressure stops trending up.

Sample ID	Cycle	Set Pressure (PSI)	Fill Time (sec)	Transfer Pressure (PSI)	Delta P (PSI)
Initial Setting	3				
(30% of max) psi	4				
	5				
(Initial + 5%) psi	3				
	4				
	5				
(Initial + 10%) psi	3				
	4				
	5				
(Initial + 15%) psi	3				
	4				
	5				
(Initial + 20%) psi	3				
	4				
	5				
(Initial + 25%) psi	3				
	4				
	5				
(Initial + 30%) psi	3				
	4				
	5				
(Initial + 35%) psi	3				
	4				
	5				
(Initial + 40%) psi	3				
	4				
	5				
(Initial + 45%) psi	3				
	4				
	5				
(Initial + 50%) psi	3				
	4				
	5				
(Initial + 55%) psi	3				
	4				
	5				
(Initial + 60%) psi	3				
	4				
	5				

(NOTE 1: when fill time no longer is decreasing you should notice a gap of several hundred PSI between the set Injection Pressure value and the hydraulic pressure at transfer.)
(NOTE 2: All parts should still be short.)
(NOTE 3: At no time should the final screw cushion be 0.)

Is the process “Pressure Limited”? – Using Pressure / Velocity Graphs

This process is pressure limited. Note the <u>actual pressure</u> line meets the <u>pressure limit</u>. Also note the <u>actual velocity</u> line does not mirror the <u>set velocity</u> line.

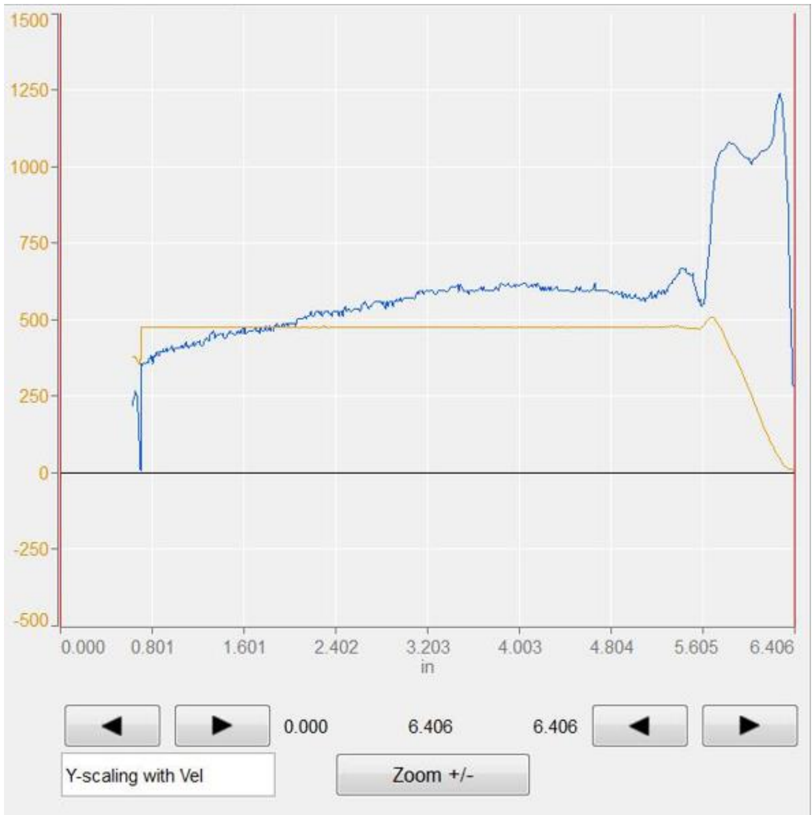


This process is not pressure limited. Note the <u>actual pressure</u> line does not meet the <u>pressure limit.</u> Also note the <u>actual velocity</u> line does mirror the <u>set velocity</u> line.

Pressure Limit (Mosaic+ Graph Method, Milacron Fan)

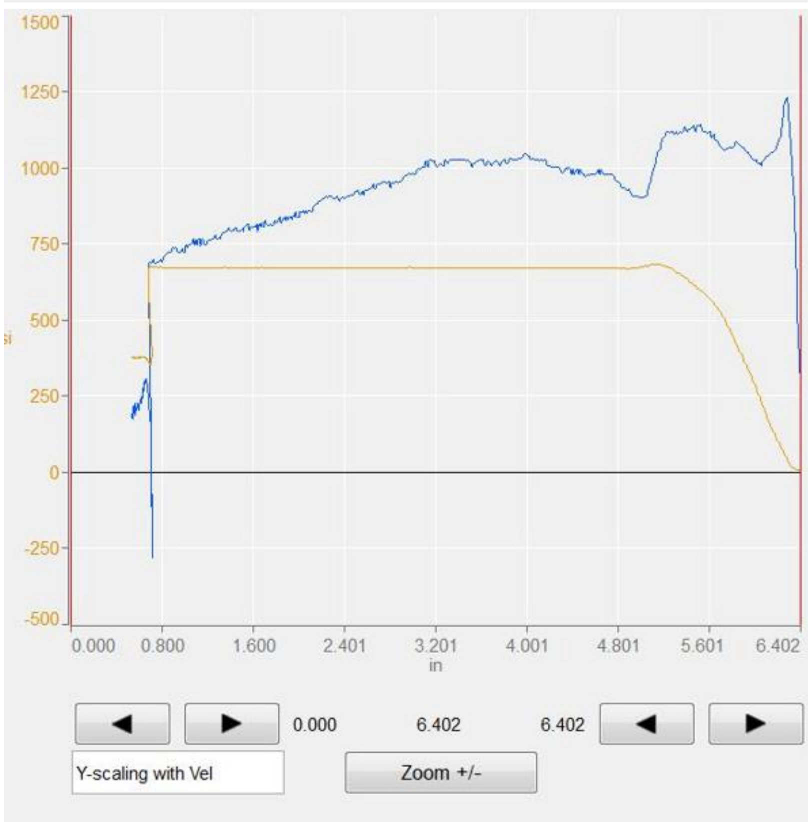
Pressure Limited

500 PSI



Pressure Limited

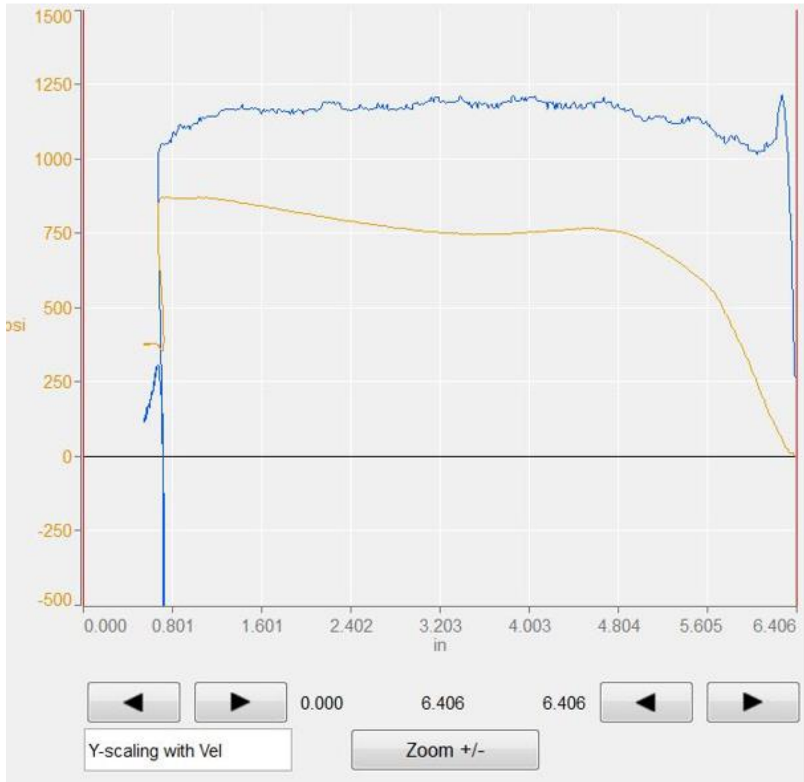
650PSI



Pressure Limit (Mosaic+ Graph Method, Milacron Fan)

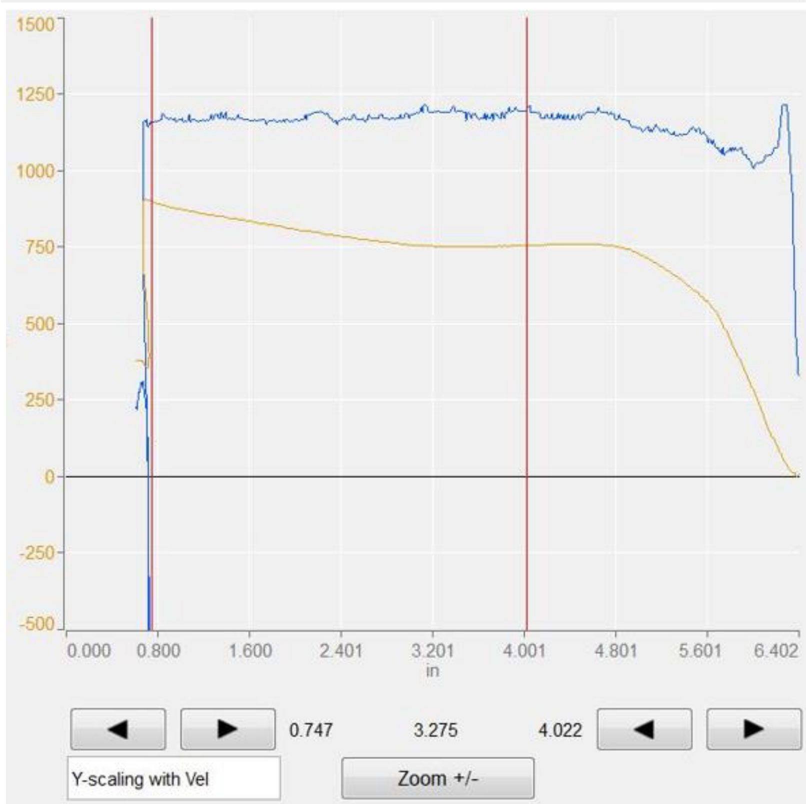
Pressure Limited

850 PSI



Not Pressure Limited

900 PSI



The IQ Experiments - Load Sensitivity

Load Sensitivity

% error = (FT1 – FT2) ÷ FT1) / (PK1 – PK2)

Low Velocity 1 in/sec

Shot Number	Melt Fill Time sec	Melt Peak Hyd. Pressure psi	Air Fill Time sec	Air Peak Hyd. Pressure psi
1	3.00	10,000	3.00	2,000
2	3.01	10,100	3.00	2,000
3	2.99	9,990	3.00	2,000
4	3.02	10,050	3.00	2,000
5	2.99	9,985	3.00	2,000
Ave	3.00	10,025	3.00	2,000

% error = 0.08%

Medium Velocity 2 in/sec

Shot Number	Melt Fill Time sec	Melt Peak Hyd. Pressure psi	Air Fill Time sec	Air Peak Hyd. Pressure psi
1	1.52	14,990	1.50	2,000
2	1.50	15,050	1.50	2,000
3	1.47	15,000	1.50	2,000
4	1.51	14,895	1.50	2,000
5	1.49	15,100	1.50	2,000
Ave	1.50	15,007	1.50	2,000

% error = 0.10%

High Velocity 3 in/sec

Shot Number	Melt Fill Time sec	Melt Peak Hyd. Pressure psi	Air Fill Time sec	Air Peak Hyd. Pressure psi
1	0.96	21,000	1.00	2,000
2	1.02	19,900	1.00	2,000
3	1.00	20,050	1.00	2,000
4	0.99	19,950	1.00	2,000
5	1.01	20,000	1.00	2,000
Ave	1.00	20,180	1.00	2,000

% error = 0.22%

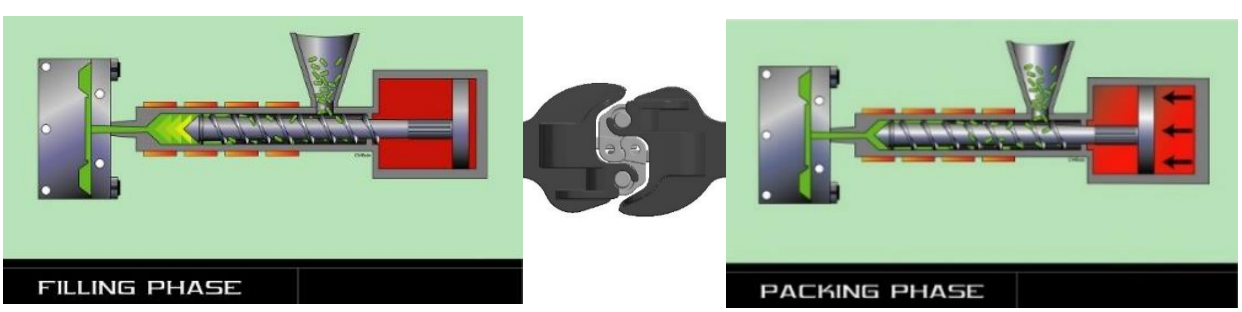
Recouple the Cycle

Re-couple the cycle by adding Pack Pressure and Hold Time

Add 2nd stage pressure. As a beginning point, set the follow-up pressures level at ½ the Pressure at Transfer. (On machines where the pressure at transfer can't be determined, add ¼ the maximum machine hold pressure).

Add a 10 second Hold Time. **These values are purely estimates.** We will adjust these settings based on the data we collect when we optimize the cycle.

> ***At this point in the set-up process, the part should appear cosmetically perfect.***
>
> ***The part dimensions may not meet customer dimensional specifications, or the physical properties might not be optimum, but the part should look complete.***
>
> ***The following three parts of the decoupled molding process will help optimize the molding cycle with regard to dimensions and physical properties.***



The Gate Seal Study

GATE SEAL STUDY

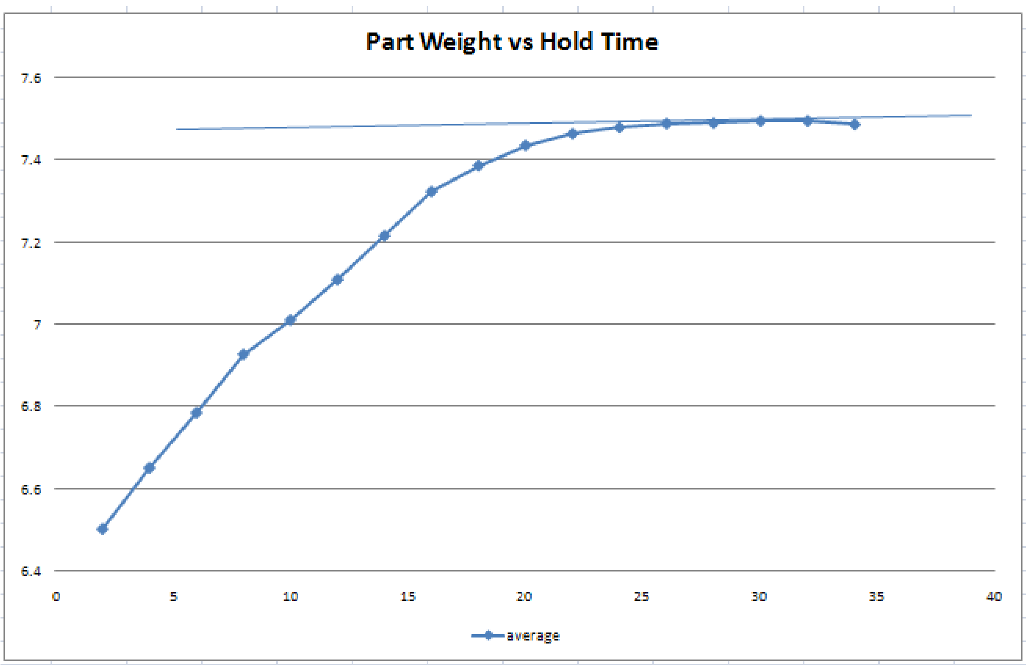
Purpose: To assure the plastic injected into the mold does not leak back out.

Method: Weigh parts to see if material has leaked out as hold time is increased.

Procedure:

1. Increase hold time, beginning decoupled at 0 sec. Observe when a full part is produced.
2. Increase hold time by a few seconds and weigh the parts produced. (1 sec for small parts, 2 sec for larger parts, 3 sec for huge parts.) Reduce cooling time so the cycle time remains constant.
3. Repeat step 2 until no more weight gain is observed.
4. Graph the data.

Set Hold Time (sec)	Average Part Weight (g)
30	22.8730
28	22.9629
26	23.0167
24	22.9073
22	22.6327
20	22.6760
18	22.8155
16	22.5689
14	22.7038
12	22.7753
10	22.4547
8	22.2632
6	22.0229
4	21.5232
2	21.0096



Part Weight vs. Hold Time

Cooling Time (sec)	Hold Time (sec)	Sample 3 Part Weight	Sample 4 Part Weight	Sample 5 Part Weight	Part Weight Average

Some commonsense discussion on hold time and gate seal:

Cycle development using Decoupled 2 and Scientific Molding, all employ a weight loss method for determining part gate seal and hold time. Typically, the correct hold time is established at 110% of the point where parts begin to lose weight because the gate has not sufficiently sealed. An alternate method for determining hold time and gate seal is possible if the mold is instrumented with a cavity pressure sensor. An analysis of cavity pressure will show a drop in cavity pressure due to gate leakage if the hold time was too short and the gate didn't have time to freeze. This is the method used in Decoupled 3 Molding.

Regardless which of these two methods is used several commonsense observations can be made about establishing hold time and gate seal.

Observation #1 – No gate seal equals lower part quality.

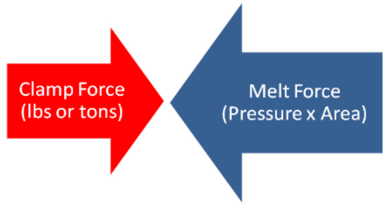
Consider this. Frequently parts are molded on a cycle where hold time is set too short. The motivation behind this setting is to produce a quicker and more profitable molding cycle. Assume the part produced on this shorter hold time has been correctly filled and packed. Material was then allowed to leak out of the non-sealed gate. How much? The actual amount of leakage will vary from cycle to cycle. In fact, it is variability that is the problem. Variable part weight is directly representative of variable dimensions. A process established with a hold time to short will be less capable than a process established with hold time set correctly. Quality goes down.

Observation #2 – Cheating on hold time doesn't reduce cycle time?

Consider this. By design the gate of an Injection Molded part is one of the thinnest cross sections in the part. Consequently, the gate will always cool before other sections of the part. For those other sections to cool prior to ejection we need to establish a cooling time greater than our gate freeze (gate seal) time. If we cheat on gate seal time, we usually need to wait for cooling time to expire! If the screw recovers and sits while preparing the next shot, the gate seal time could have been longer without any cycle time penalty!

Set a Pack Pressure

Set an Initial Pack Pressure

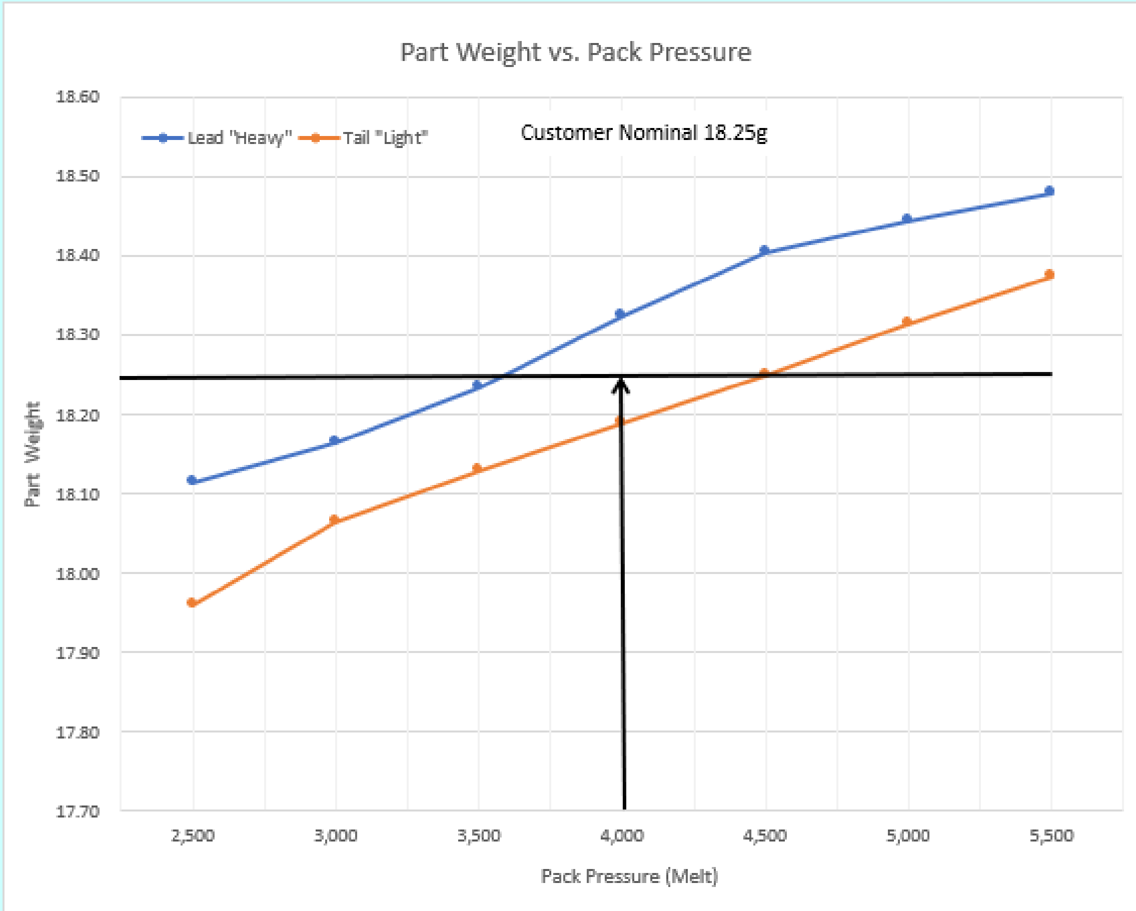


Flash
(Cavity Pressure
Overcomes Clamp Force)

No Fill
(Not enough pressure
for 2-5% of fill)

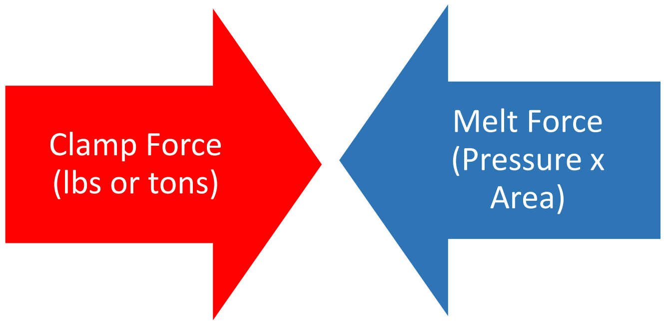
Pack Setting (PSI)	Cavity Pressure (PSI)	Dimension 1	Dimension 2	Dimension 3
		OD	OAL	Hole Ø
		(0.001 in.)	(0.001 in.)	(0.001 in.)
No Fill				
Flash				

Controlling Dimensions using Pack Pressure

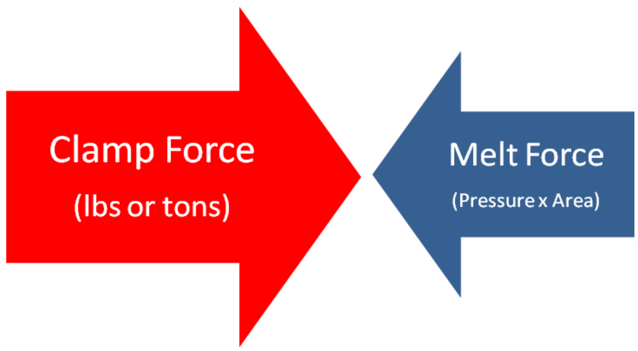


Plastics Black Belt	Ave		Measured Weight			
Pack Pressure	Heavy Part 1	Light Part 2	Heavy Part 1	Heavy Part 2	Light Part 1	Light Part 2
2,500	18.12	17.96	18.10	18.13	17.94	17.98
3,000	18.17	18.07	18.16	18.17	18.05	18.08
3,500	18.24	18.13	18.23	18.24	18.14	18.12
4,000	18.33	18.19	18.32	18.33	18.19	18.19
4,500	18.41	18.25	18.41	18.40	18.22	18.28
5,000	18.45	18.32	18.43	18.46	18.31	18.32
5,500	18.48	18.38	18.47	18.49	18.37	18.38

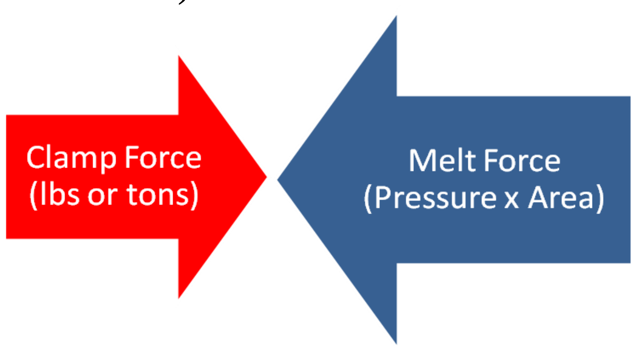
Why does a mold flash?



On each cycle of an Injection Molding machine, the melt cavity pressure during packing attempts to defeat the clamp and force open the mold.

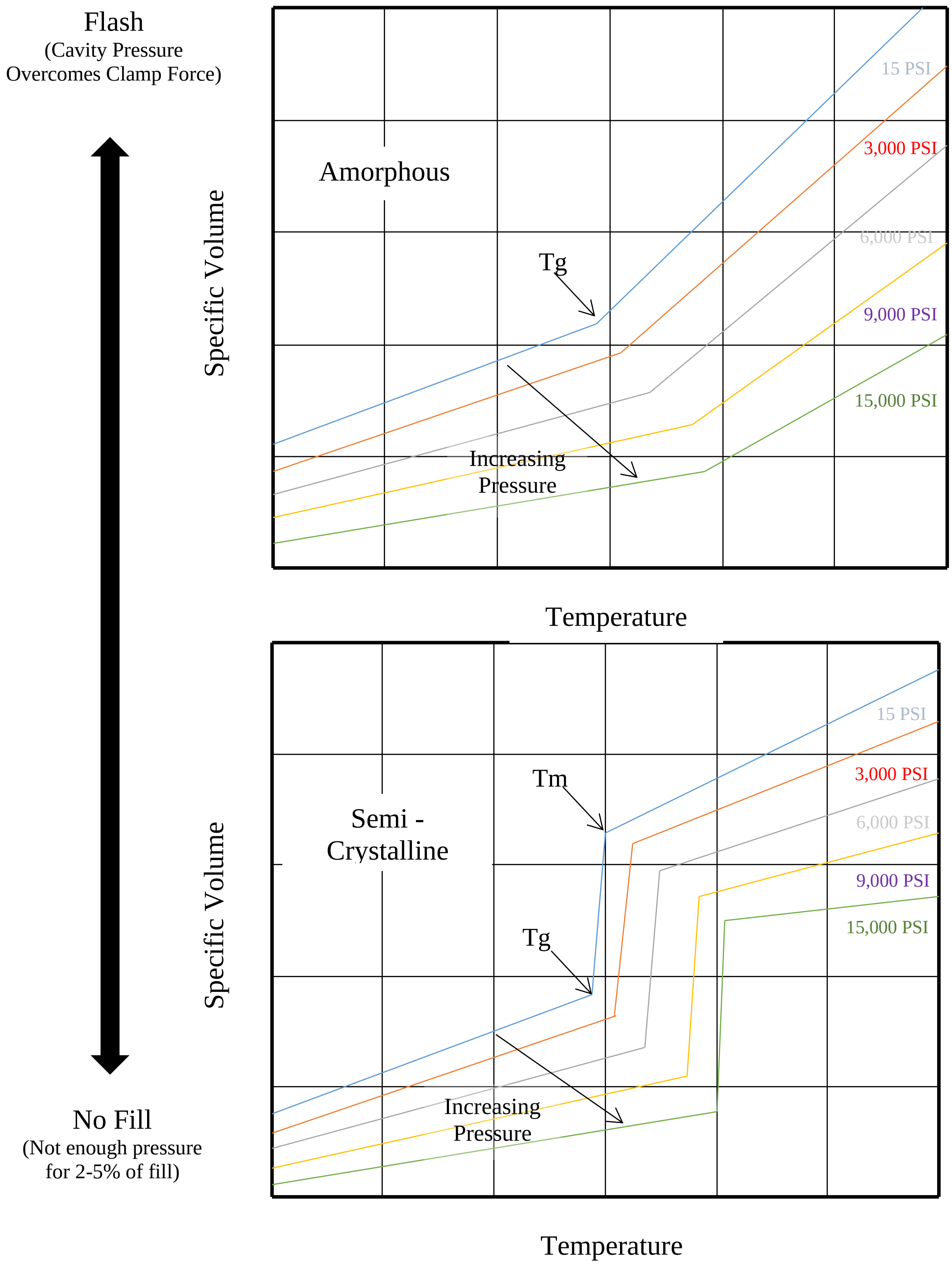


If the clamp force is greater than the melt force during packing the clamp wins the battle (red arrow wins) and flash will not occur.

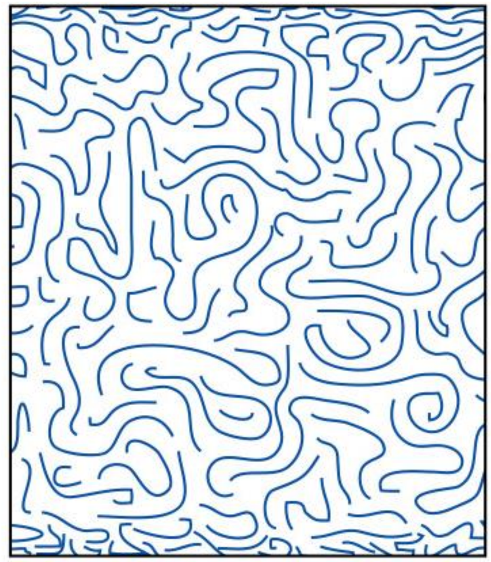


If the melt force in the cavity during packing is greater than the clamp force (blue arrow wins), we observe flash in the molded parts.

Center the Process Using Pack Pressure (Cavity Pressure)



Cavity Filling and Part Dimensions - Amorphous Materials

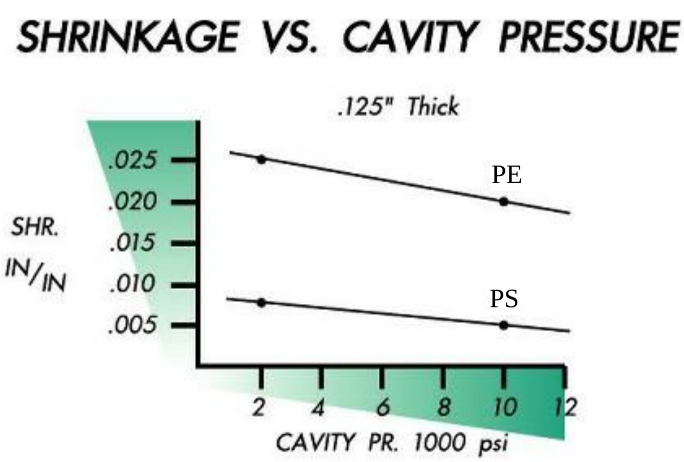


Orientation Stress

Orientation Stress is stored in Molecules. These molecules would prefer to return to a random coil but are frozen with **Residual Internal Orientation Stress**.

Packing Stress

Packing Stress is stored in Molecules. These molecules would prefer to be spaced exactly according to the needed free volume for a given temperature. Some are frozen to close, in compression. Others are frozen too far apart, in tension. All yield **Residual Internal Packing Stress**.

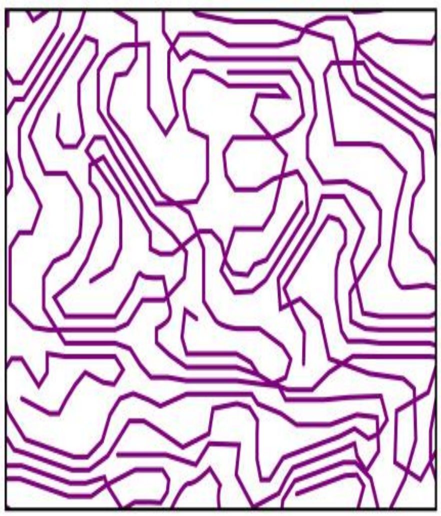


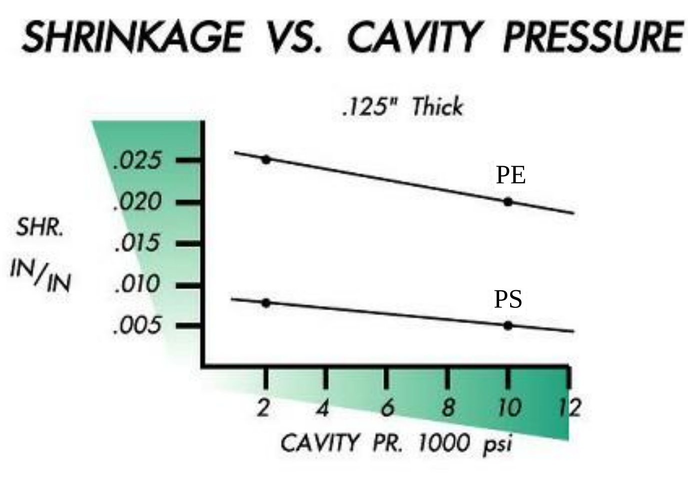
Polystyrene	Polyethylene
2,000 psi = 0.007 in/in 10,000 psi = 0.004 in/in	2,000 psi = 0.025 in/in 10,000 psi = 0.017 in/in

Cavity Filling and Part Dimensions - Crystalline Materials

% Crystallinity

In addition to Shrinkage Due to Orientation Stress and Packing Stress Crystalline materials also shrink because of crystallinity.

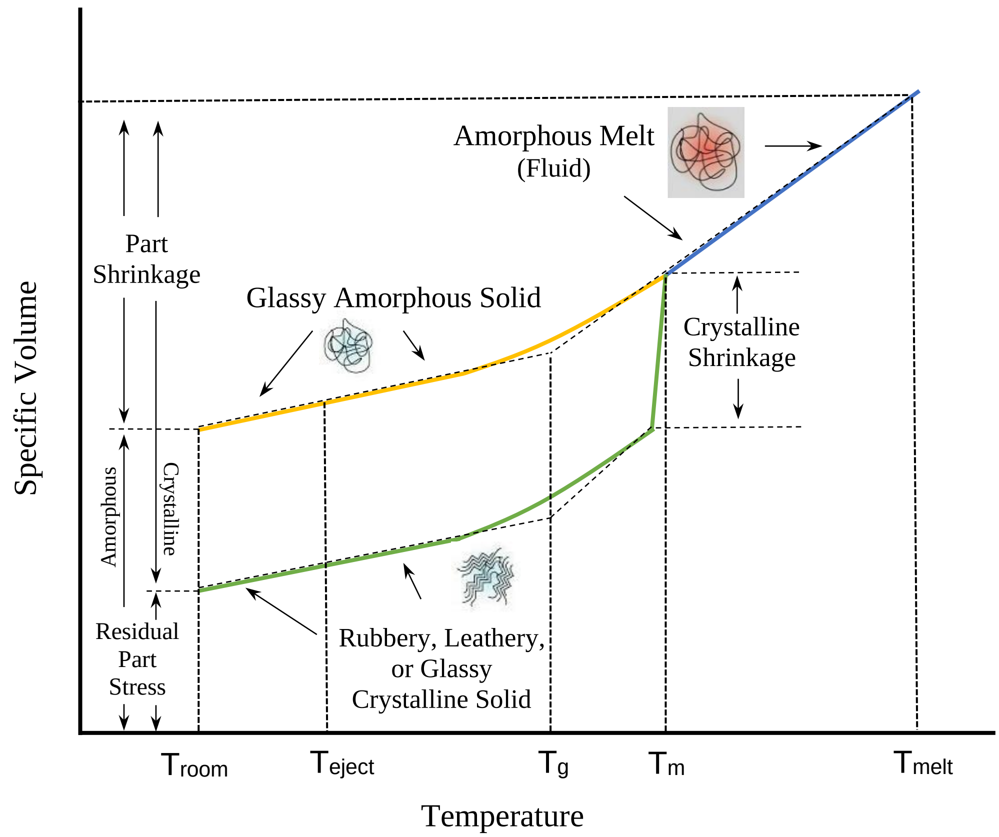




Polystyrene	Polyethylene
2,000 psi = 0.007 in/in	2,000 psi = 0.025 in/in
10,000 psi = 0.004 in/in	10,000 psi = 0.017 in/in

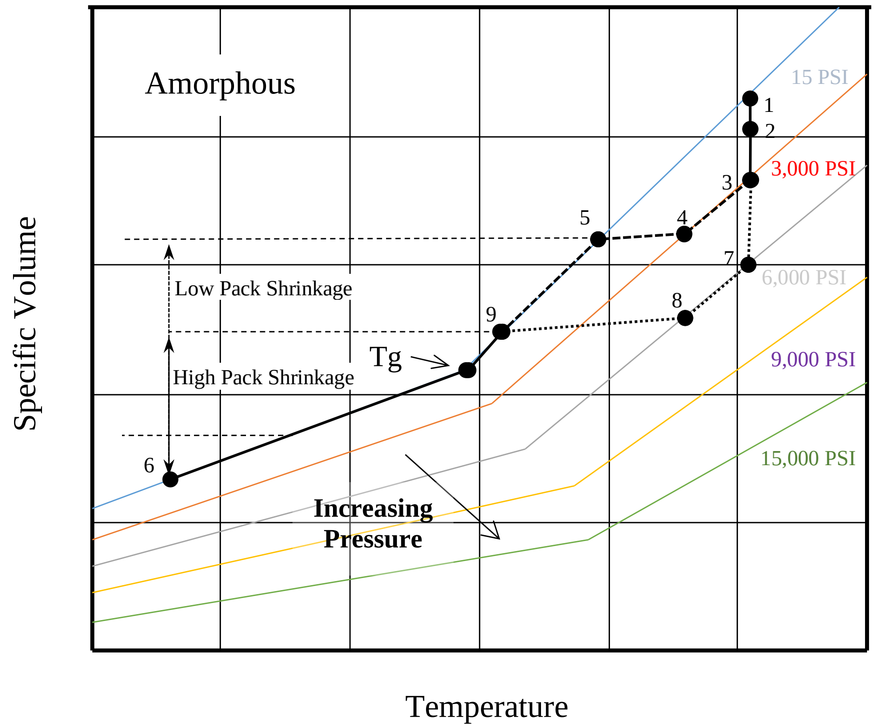
Graphing Amorphous and Crystalline Shrinkage

As mentioned earlier amorphous and crystalline materials shrink very differently.



PvT Graphs

We can use a PvT (Pressure-volume-Temperature) Diagram to show how each part of the injection molding cycle impacts shrinkage.



Low Pack Scenario
1: Injection Begins (Atmospheric Pressure)
1-2: Fill (Hagen-Poiseuille pressure rises with increasing L)
2: Transfer (Pressure at Transfer)
2-3: Packing
3: Low Pack Pressure
3-4: Low Pack Holding
4: Low Pack Gate Seal (Gate Freeze)
4-5: Low Pack Part Cooling
5: Shrinkage is enough that low pack cavity pressure reaches atmospheric pressure
6: Part ejection

High Pack Scenario
1: Injection Begins (Atmospheric Pressure)
1-2: Fill (Hagen-Poiseuille pressure rises with increasing L)
2: Transfer (Pressure at Transfer)
2-7: Packing
7: High Pack Pressure
7-8: High Pack Holding
8: High Pack Gate Seal (Gate Freeze)
8-9: High Pack Part Cooling
9: Shrinkage is enough that high pack cavity pressure reaches atmospheric pressure
6: Part ejection
.

Controlling Dimension – Comparison of Polymer Materials

When actual shrinkage in polymer materials is observed it becomes apparent that the three sources of shrinkage, we have discussed don't all have equal power.

As a rule of thumb (which means it's kind of right and always wrong):

Orientation Stress (Strength = 1x)
(only found in non-Newtonian materials)

Packing Stress (Strength = 2x)
(found in all materials)

Crystalline Shrinkage (Strength = 10x)
(only found in crystalline materials)

Based on the observations, which polymer materials are easiest to mold with little dimensional variability?

Based on the observations, which polymer materials are hardest to mold with little dimensional variability?

Meeting Critical Dimensions

Meeting Critical Dimensions

	Shrinkage from:			Ease of Controlling Dimensions
	Packing	Orientation	Crystallinity	
Newtonian Amorphous	Yes	No	No	**Very easy** Packing pressure controls part size.
Non-Newtonian Amorphous	Yes	Yes	No	**Harder** Shrinkage is controlled by packing pressure, injection velocities, and cooling rate
Semi-Crystalline	Yes	Yes	Yes	**Challenging** Shrinkage is developed by all three factors. Cooling rate plays a major role too.

Meeting tight tolerance may be very difficult with certain types of polymers and relatively easy with others.

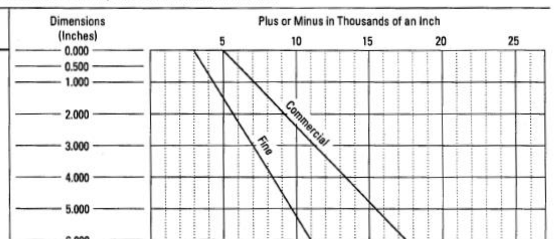
How tight of a tolerance is reasonable for any material? SPI published a guide called" Standards and Practices of Plastic Molders" as a starting point to referee the discussion and assignment of tolerances between molders and their customers.

Standards & Practices of Plastics Molders

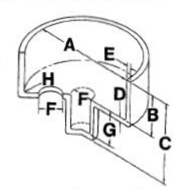
Material Polypropylene (PP)

Note: The *Commercial* values shown below represent common production tolerances at the most economical level. The *Fine* values represent closer tolerances that can be held but at a greater cost. Any addition of fillers will compromise physical properties and alter dimensional stability. Please consult the manufacturer.

Drawing Code: A = Diameter (See note #1); B = Depth (See note #3); C = Height (See note #3)



Drawing Code	Dimensions (Inches)	Comm. ±	Fine ±
	6.000 to 12.000 for each additional inch add (inches)	0.005	0.003
D = Bottom Wall	(See note #3)	0.006	0.003
E = Side Wall	(See note #4)	0.006	0.003
F = Hole Size Diameter (See note #1)	0.000 to 0.125	0.003	0.002
	0.126 to 0.250	0.004	0.003
	0.251 to 0.500	0.005	0.004
	0.501 & over	0.008	0.006
G = Hole Size Depth (See note #5)	0.000 to 0.250	0.005	0.003
	0.251 to 0.500	0.006	0.004
	0.501 to 1.000	0.009	0.006
H = Corners, Ribs, Fillets	(See note #6)	0.029	0.016
Flatness (See note #4)	0.000 to 3.000	0.022	0.014
	3.001 to 6.000	0.036	0.021
Thread Size (Class)	Internal	1	2
	External	1	2
Concentricity	(See note #4) (F.I.M.)	0.015	0.012
Draft Allowance Per Side	(See note #5)	1.5°	0.5°
Surface Finish	(See note #7)		
Color Stability	(See note #7)		



REFERENCE NOTES

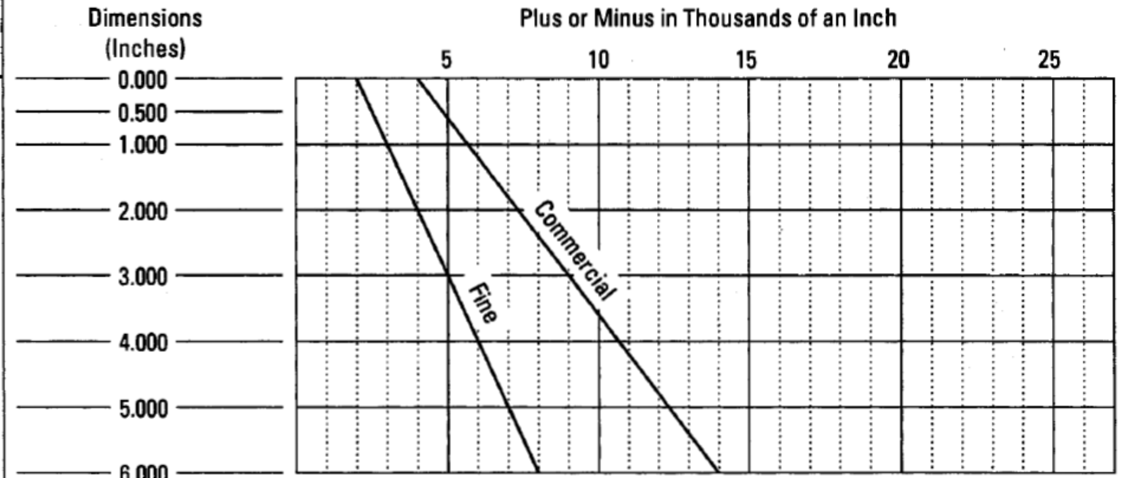
1. These tolerances do not include allowance for aging characteristics of material.
2. Tolerances are based on 0.125 inch wall section.
3. Parting line must be taken into consideration.
4. Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is sometimes impossible to achieve. Walls of non-uniform thickness should be gradually blended from thick to thin.
5. Care must be taken that the ratio of the depth of a cored hole to its diameter does not reach a point that will result in excessive pin damage.
6. These values should be increased whenever compatible with desired design and good molding techniques.
7. Customer-Molder understanding is necessary prior to tooling.

Standards & Practices of Plastics Molders

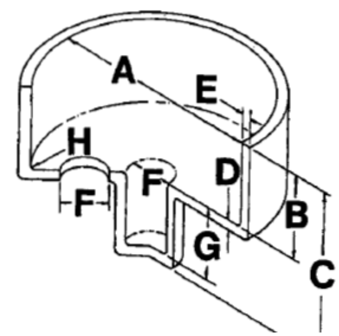
Material
Polyoxymethylene
(Acetal) (POM)

Note: The *Commercial* values shown below represent common production tolerances at the most economical level. The *Fine* values represent closer tolerances that can be held but at a greater cost. Any addition of fillers will compromise physical properties and alter dimensional stability. Please consult the manufacturer.

Drawing Code	Dimensions (Inches)	Plus or Minus in Thousands of an Inch	
A = Diameter (See note #1) B = Depth (See note #3) C = Height (See note #3)	0.000 to 6.000 (see chart)		



Drawing Code	Dimensions (Inches)	Comm. ±	Fine ±
	6.000 to 12.000 for each additional inch add (inches)	0.004	0.002
D = Bottom Wall	(See note #3)	0.004	0.002
E = Side Wall	(See note #4)	0.004	0.002
F = Hole Size Diameter (See note #1)	0.000 to 0.125	0.002	0.001
	0.126 to 0.250	0.003	0.002
	0.251 to 0.500	0.004	0.002

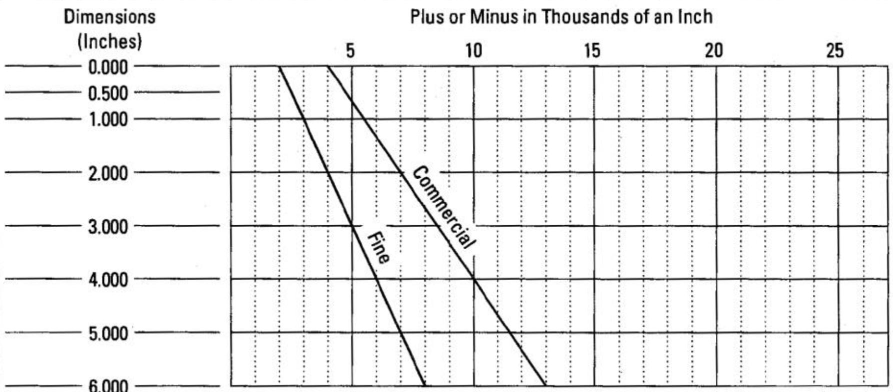


Determining Reasonable Specifications

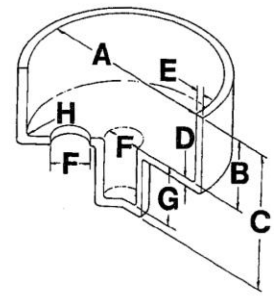
Standards & Practices of Plastics Molders	**Material** Acrylic

Note: The *Commercial* values shown below represent common production tolerances at the most economical level. The *Fine* values represent closer tolerances that can be held but at a greater cost. Any addition of fillers will compromise physical properties and alter dimensional stability. Please consult the manufacturer.

Drawing Code	Dimensions (Inches)	Plus or Minus in Thousands of an Inch
A = Diameter (See note #1) B = Depth (See note #3) C = Height (See note #3)		

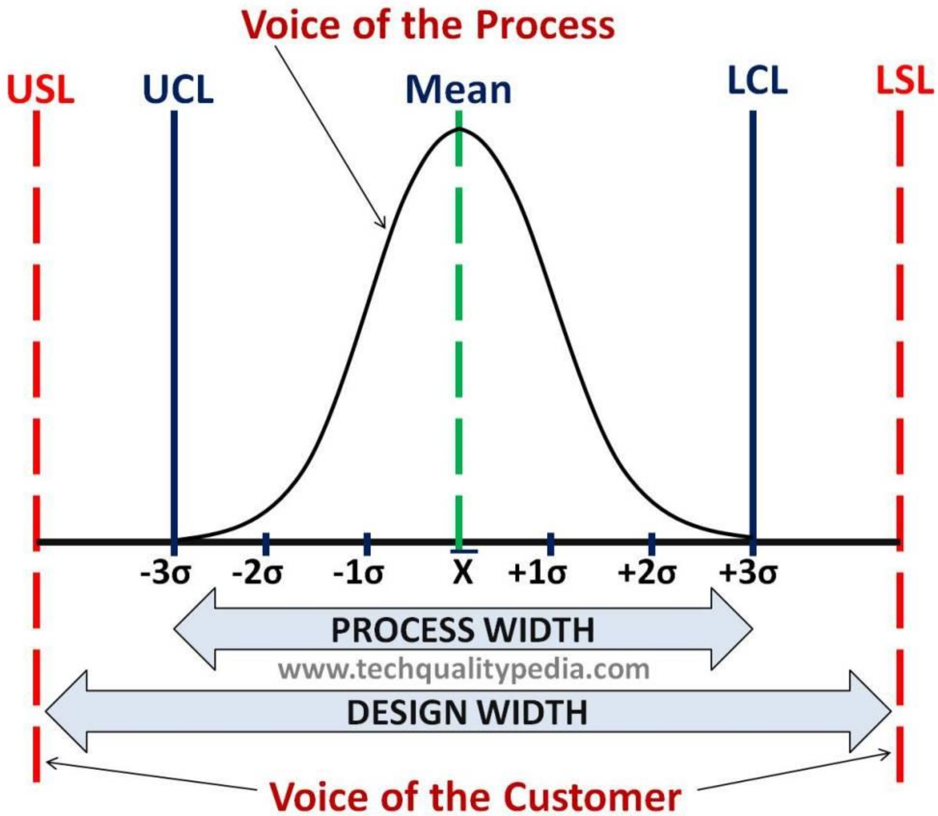


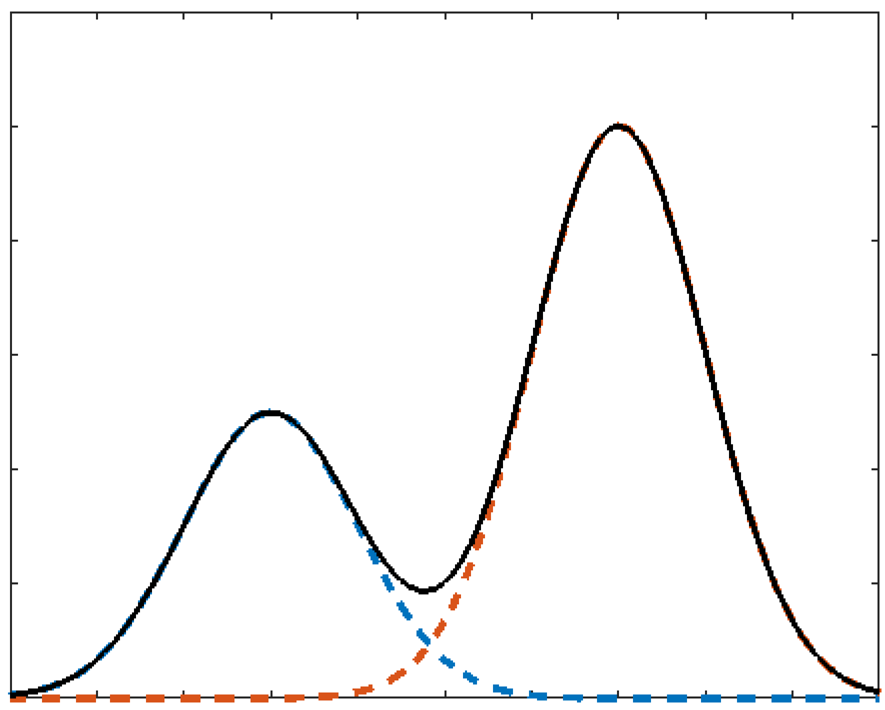
Drawing Code	Dimensions	Comm. ±	Fine ±
	6.000 to 12.000 for each additional inch add (inches)	0.004	0.002
D = Bottom Wall	(See note #9)	0.005	0.002
E = Side Wall	(See note #2)	0.004	0.002
F = Hole Size Diameter (See note #1)	0.000 to 0.125	0.003	0.001
	0.126 to 0.250	0.003	0.002
	0.251 to 0.500	0.004	0.002
	0.501 & over	0.005	0.003
G = Hole Size Depth (See note #5)	0.000 to 0.250	0.004	0.002
	0.251 to 0.500	0.004	0.002
	0.501 to 1.000	0.006	0.003
H = Corners, Ribs, Fillets	(See note #6)	0.025	0.012
Flatness	0.000 to 3.000	0.013	0.008
(See note #4)	3.001 to 6.000	0.023	0.015
Thread Size (Class)	Internal	1	2
	External	1	2
Concentricity	(See note #4) (F.I.M.)	0.010	0.006
Draft Allowance Per Side	(See note #5)	1.5°	0.75°
Surface Finish	(See note #8)		
Color Stability	(See note #7)		

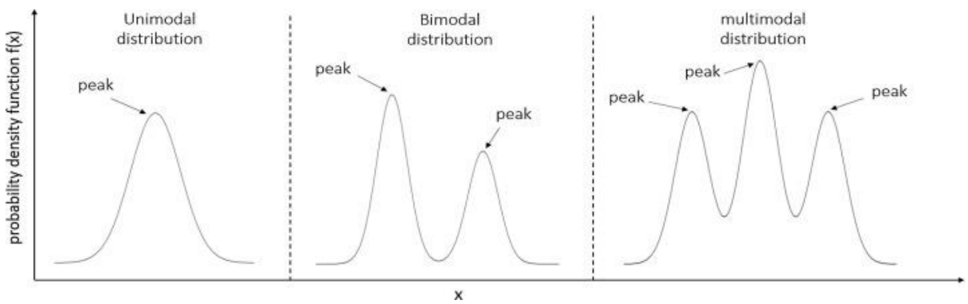


Reference Notes

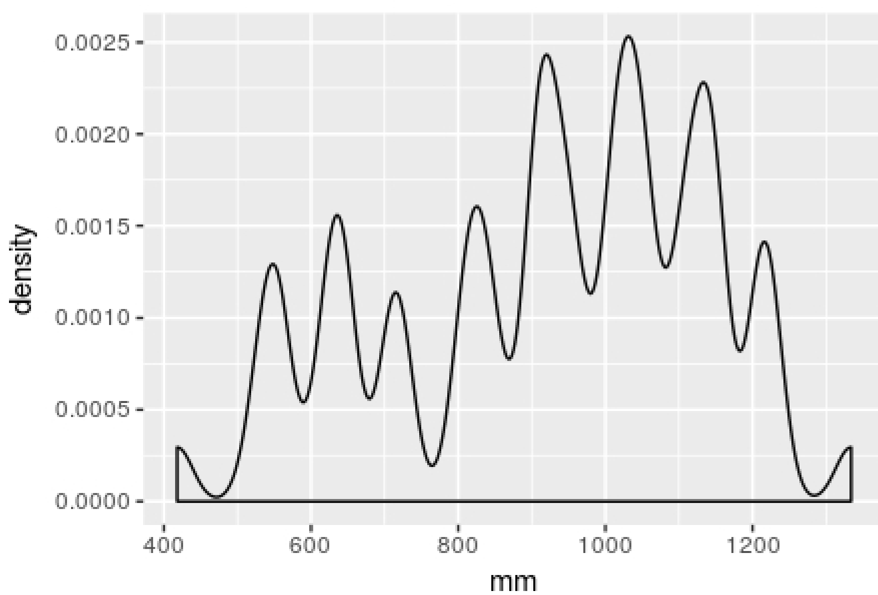
1. These tolerances do not include allowance for aging characteristics of material.
2. Wall thickness should be as uniform as possible.
3. Parting line must be taken into consideration.
4. Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is sometimes impossible to achieve. Walls of non-uniform thickness should be gradually blended from thick to thin.
5. Care must be taken that the ratio of the depth of a cored hole to its diameter does not reach a point that will result in excessive pin damage.
6. Large radius is desirable to minimize part breakage.
7. Customer-Molder understanding is necessary prior to tooling.
8. Part surface finish is dependent on mold finish.
9. Based on nominal 0.125 inch wall.


Voice of the Process
USL
UCL
Mean
LCL
LSL
-3σ
-2σ
-1σ
X
+1σ
+2σ
+3σ
PROCESS WIDTH
www.techqualitypedia.com
DESIGN WIDTH
Voice of the Customer

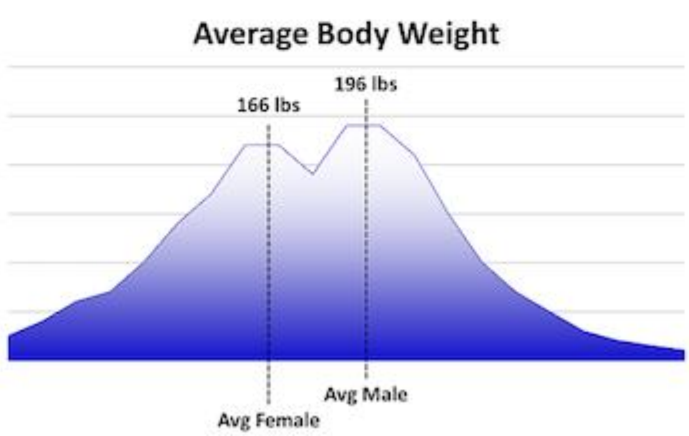




probability density function f(x)
Unimodal distribution
peak
Bimodal distribution
peak
peak
multimodal distribution
peak
peak
peak
x

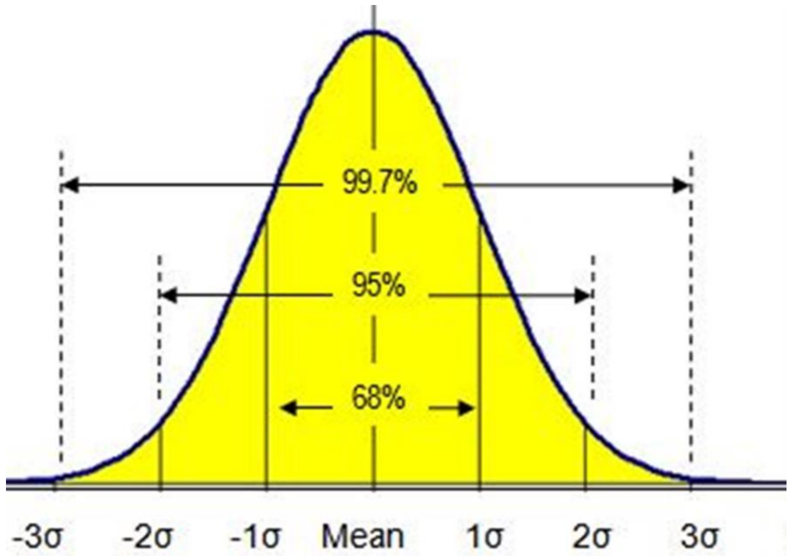


0.0025
0.0020
0.0015
0.0010
0.0005
0.0000
density
400
600
800
1000
1200
mm



Average Body Weight
196 lbs
166 lbs
Avg Male
Avg Female

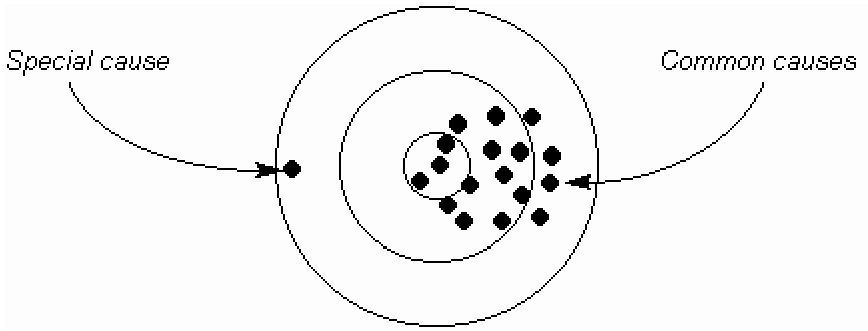
Variation and Predictability



C_{pk}	"X"-Sigma Quality	% Out-of-Specification ①	PPM Out-of-Specification ①
0.33	1	31.7%	~317,300
0.50	1 ½	13.4%	~133,600
0.67	2	4.6%	~45,500
0.83	2 ½	1.24%	~12,400
1.00	3	0.27%	~2,700
1.17	3 ½	0.047%	~465
1.33	4	0.0063%	~63
1.50	4 ½ ②	6.8×10^{-4} %	6.8
1.67	5	5.7×10^{-5} %	0.57
1.83	5 ½	3.8×10^{-6} %	0.038
2.00	6 ②	2.0×10^{-7} %	0.002
2.33	7	2.6×10^{-10} %	0.0000026
2.67	8	1.2×10^{-13} %	0.0000000012

Controlling Variation

Common and Assignable (Special) Causes of Variation



"In Control" vs. "Out of Control"

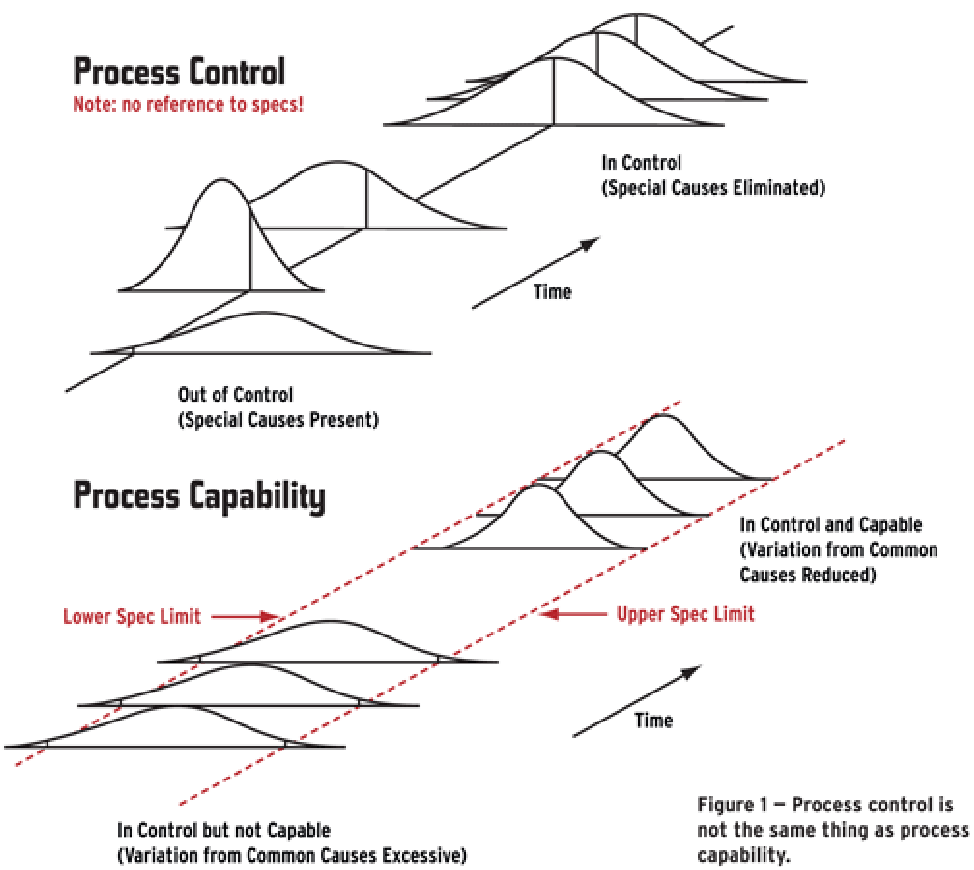
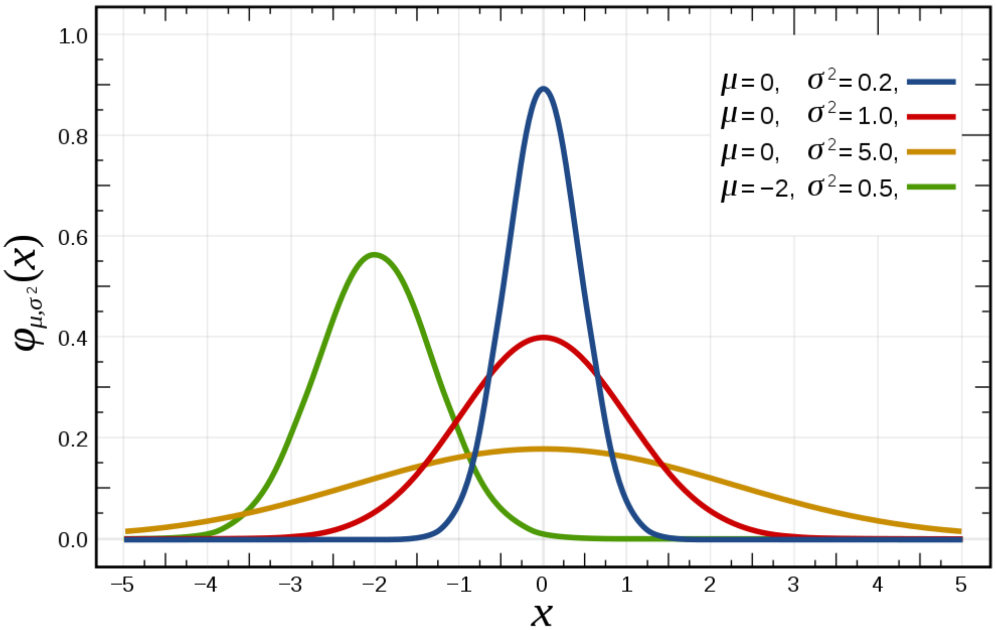


Figure 1 – Process control is not the same thing as process capability.

Optimizing an In-Control Process



Optimizing an Out-of-Control Process

GOOD LUCK!

Milacron

Processing 1

Control Utilization and Cycle Development

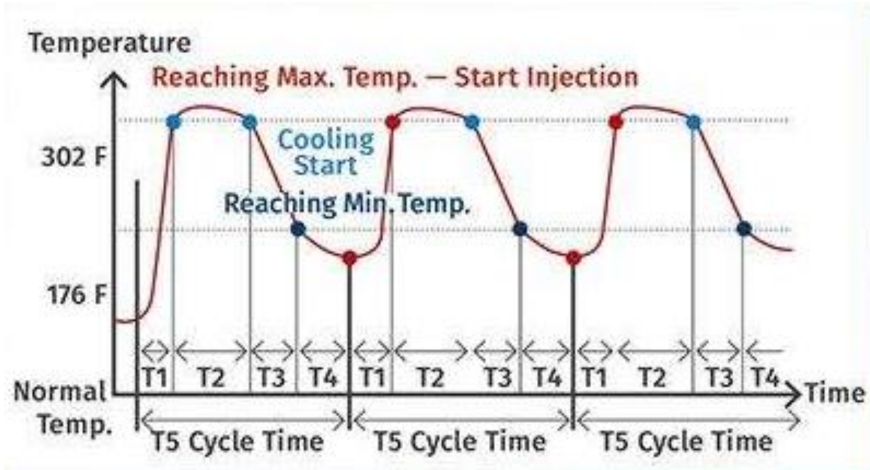
Day 3

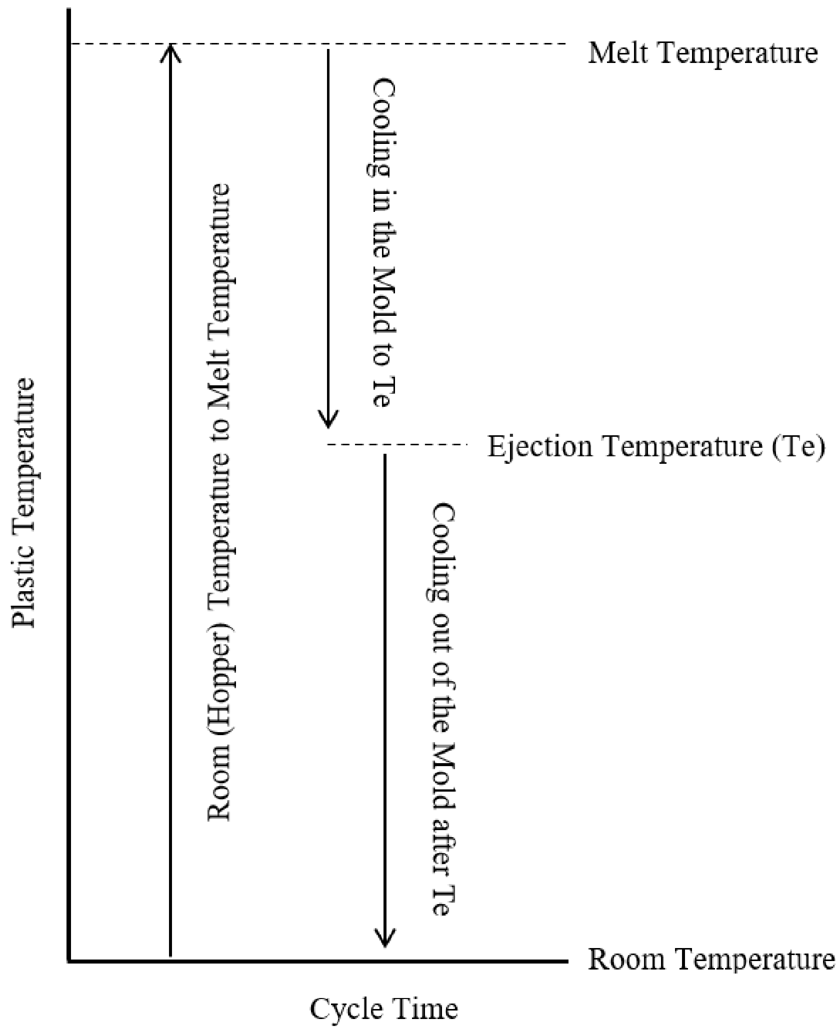
Cycle Optimization

Thursday Cycle Optimization

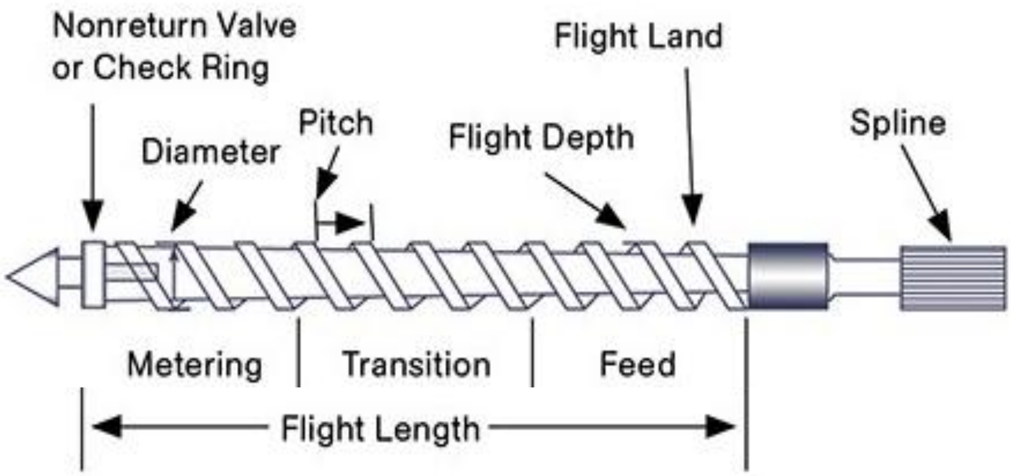
7:30 a.m.	Check in with Refreshments
8:00 a.m.	- Quick Review with Q & A
	- Barrel Temperatures
	- Temperature Profiles
	Preview of Tacking Temperature Experiment
	Lab Session 9: Tacking Temperature
	Preview of Tacking Temperature Experiment
9:45 a.m.	**Break/Networking**
10:00 a.m.	Preview of 30-30-30 Melt Temperature Experiment
	Lab Session 10: The 30-30-30 Melt Temperature
	Review 30-30-30 Melt Temperature Data
	- Cooling, Delta T, and Ejection Temperature
	- Runner Balance
	- Cooling Balance
	- a Balance of Fill Example
	Preview Balance of Fill analysis
	Lab Session 8: Mold Balance of Fill Parts
	Review Balance of Fill Data
Noon	**Lunch**
12:30 p.m.	- Tying it all together
	Cushion, Shot Size, Transfer, Suck Back
	Injection Fill Pressure Limit
	Injection Speed(s)
	Screw Speed, Backpressure, Barrel Heats
	Coolant Temperature and Flow Rate, Cooling Time
	Pack Pressure
	Hold Time
	Final Q&A
2:00 p.m.	Adjourn

Heading and Cooling - Delta T





The Injection Molding Screw



- Feed Zone (approximately 6 turns)
  (Preheat the material, feed material to the compression zone)

  •Deepest Flights
  •Conveys pellets to the compression zone
  •Begins heating and melting
  •Forces air back toward feed throat
- Compression Zone ((approximately 12 turns)
  (Apply shear and heat, converting pellets to a melt)

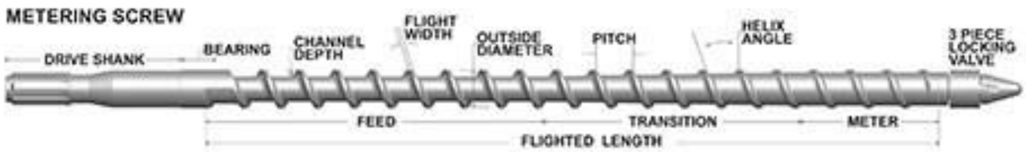
  •Steady reduction in flight depth
  •80% of melt development happens here
  •Significant heat from friction (shear)
  •Also important site of mixing
- Metering Zone (approximately 6 turns)
  (Homogenize the melt for both temperature and composition)

  •Shallow and constant flight depth
  •Does little to change temperature
  •Final mixing occurs here

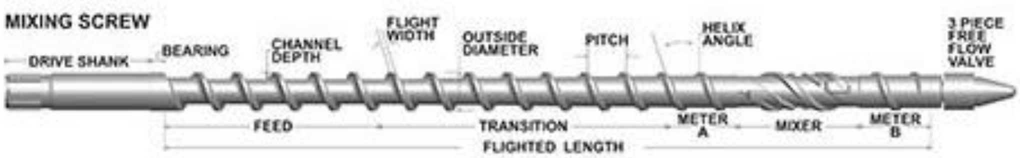
Screw Design

Special Purpose Screws

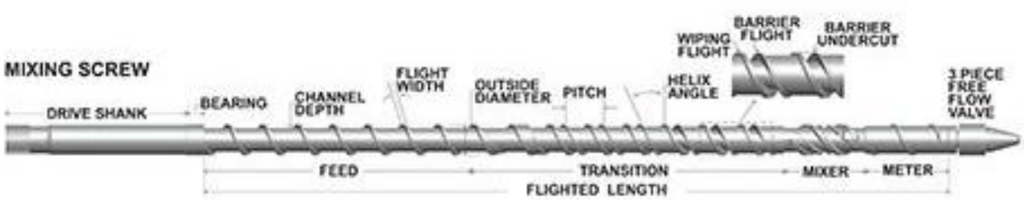
The first screw shown below is a general-purpose metering screw similar to the one just described. It is adequate for most Injection Molding applications.



If more mixing is needed one option is a mixing screw with a mixer element built into the screw design.



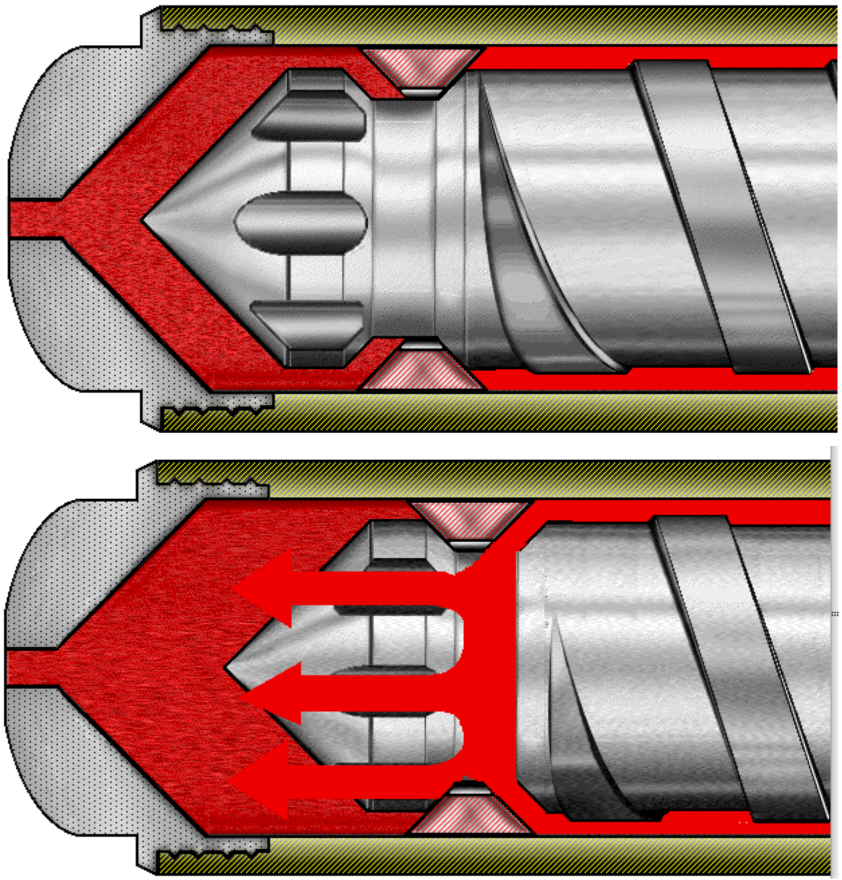
Another option for providing more mixing is a screw with barrier flights.



In Injection Molding, screw design may be necessary if mixing using screw speed and back-pressure aren’t enough.

Check (nonreturn) Valve

Check (nonreturn) Valve (controls the flow of plastic over the screw during injection and screw recovery)



Types

Ball Check

Slider Ring

Over time a check valve will wear. The amount of wear can be measured by observing if the screw continues to drift forward after full pack is achieved using the Static Check Ring Experiment and by seeing if the ring is leaking during Injection using the Dynamic Check Ring Experiment.

Static Check Ring Experiment

1. Set up a recoupled cycle with pack pressure set just below flash.

2. Set the hold time to Cooling time – Screw Recovery Time. (must be more than gate seal).

3. For 5 cycles, record the actual screw position at set intervals after transfer.

4. Plot the average position at each time interval.

5. Record the shortest time where the cushion does not change.

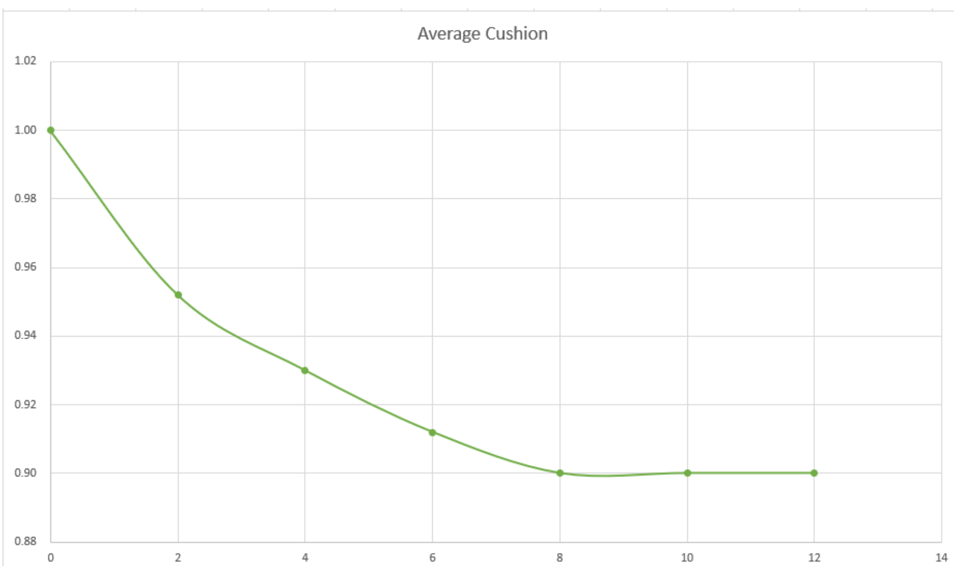
6. The variance between the minimum and maximum cushion should be less than 15%.

Variation = ((Maximum Cushion – Minimum Cushion) / Average Cushion) x 100

NOTE 1: If the screw never stops moving it is an indication the check ring is not sealing.

NOTE 2: The Variation should be recorded over time as an indication of check ring wear.

	Screw Position (Seconds after Position Transfer)						
	0	2	4	6	8	10	12
Cycle 1	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Cycle 2	1.00	0.96	0.93	0.92	0.90	0.90	0.90
Cycle 3	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Cycle 4	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Cycle 5	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Average	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Minimum	1.00	0.95	0.93	0.91	0.90	0.90	0.90
Maximum	1.00	0.96	0.93	0.92	0.90	0.90	0.90
Variation	0.00%	1.05%	0.00%	1.10%	0.00%	0.00%	0.00%



The IQ Experiments - Dynamic Check Ring Experiment

Dynamic Check Ring Experiment

1. Set up a decoupled cycle with pack pressure and hold time set to 0.

2. Assure the lead part is roughly 95% full.

3. Set the fill speed to 20% of machine maximum.

4. For 5 cycles, record the short part weight for the lead cavity.

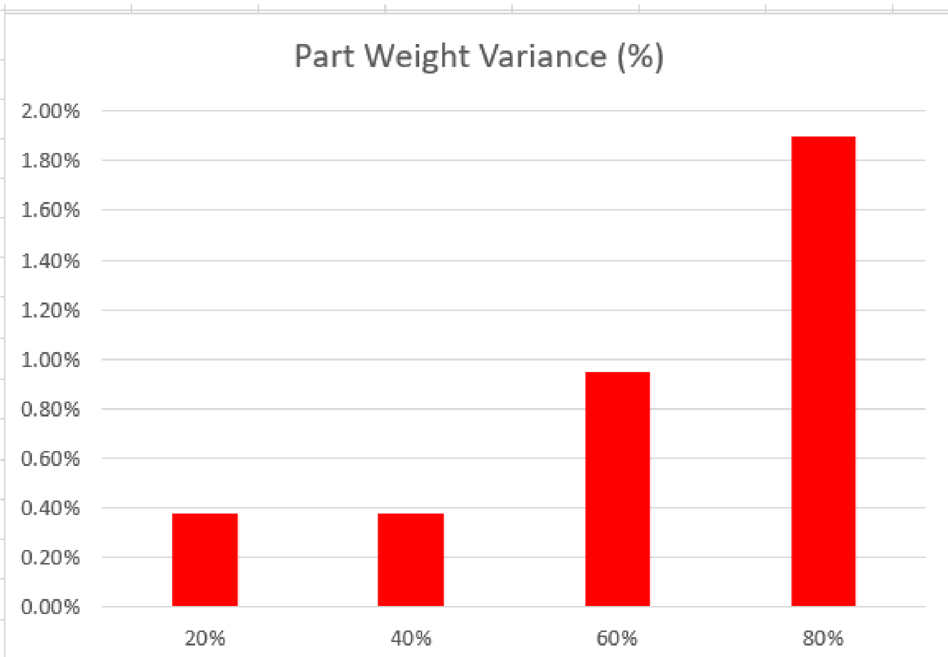
5. Repeat for Injection Speeds 40%, 60%, and 80% of the machine maximum.

5. Calculate and plot the part weight variance at each speed.

6. The variance between the minimum and maximum part weight should be less than 3%.

NOTE: The Variation should be recorded over time as an indication of check ring wear.

Inj Speed (%)	Inj Speed (in/sec)	Part Weight								
		Cycle								
		1 (g)	2 (g)	3 (g)	4 (g)	5 (g)	Ave (g)	Min (g)	Max (g)	Variance (g)
20%	1.00	5.26	5.27	5.26	5.28	5.28	5.27	5.26	5.28	0.38%
40%	2.00	5.26	5.27	5.26	5.28	5.28	5.27	5.26	5.28	0.38%
60%	3.00	5.24	5.27	5.26	5.29	5.28	5.268	5.24	5.29	0.95%
80%	4.00	5.21	5.31	5.28	5.22	5.27	5.258	5.21	5.31	1.90%

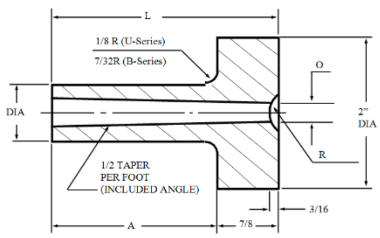


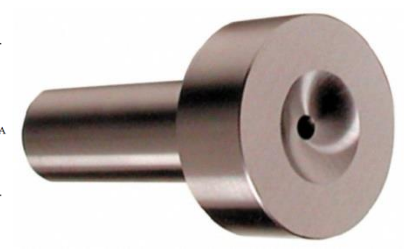
Nozzle

(Acts as a bridge from the injection unit to sprue bushing of the mold)

Nozzle Radius and Hole Size

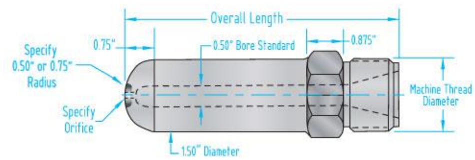
The radius and hole diameter of a nozzle must match the radius and sprue bushing on a mold. Mismatch will cause drool.





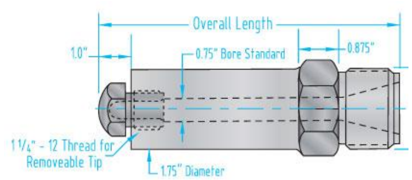
One Piece Nozzles

Many nozzles are machined as one piece. If a mold has a sprue bushing of a different radius or hole diameter the whole nozzle must be changed.



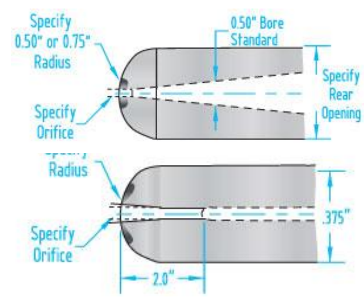
Two-Piece Nozzles

Another option is a two-piece nozzle. In this case, only the tip needs to be changed.



Taper

Materials that tend to drool need a special nozzle called a reverse taper nozzle. The nozzle on the top right is a standard nozzle. The nozzle on the bottom right is a reverse taper nozzle.

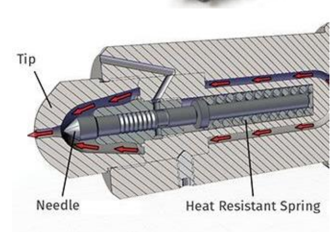


Mixing Elements

Mixing elements are sometimes added to the interior of a nozzle. Two types of mixing elements are shown at the right.

Shut-Off Nozzles

Another special-purpose nozzle is a shut-off nozzle. There are several types of shut off nozzles on the market.



Developing "The Melt"

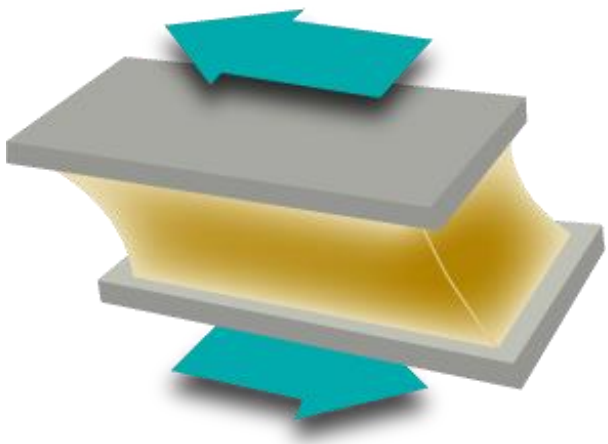
The process of converting pellets in the hopper to "the melt" in "the shot" should be focused on producing a homogeneous melt with respect to composition, temperature, and heat history.

A homogeneous melt:

Composition – perfectly mixed
Temperature – no hot spots
Heat History – all molecules have seen the same heat (temperature x time)

To achieve this homogenous melt, we try to maximize the **shear stress** placed on the pellets and melt by the mechanical action of the screw and barrel.

Shear stress (τ) is the force placed on a material when two surfaces act in opposite directions to deform the material.



In the case of an Injection Molding machine, the screw is moving, and the barrel is not producing a shearing stress.

This application of shear to produce mixing and a homogenous melt is similar to a baker kneading bread to produce homogenous dough.

The more energy input the better the mixing.

Controlling Shear on the Screw during Melting

So, how do we control the amount of shear produced when the screw runs? Two factors influence the shear energy produced:

Screw Speed

Backpressure

<u>Try this:</u>

1. Place your hands gently together.
2. Move them back and forth very slowly.
3. Gradually increase the speed of rubbing your hands.
4. As you continue to rub, suddenly press.

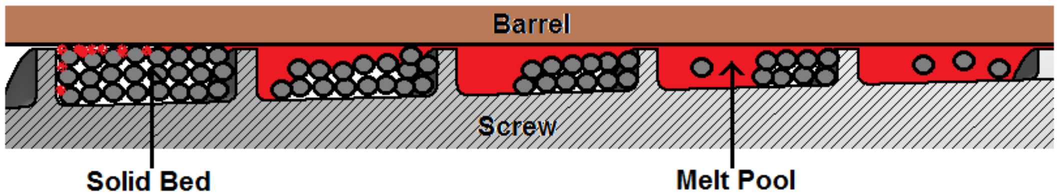
The heat you feel when you rubbed faster and faster is analogous to increasing screw speed and the press at the end is analogous to backpressure increasing the energy produced by friction and shear.

How <u>fast you rub</u> is analogous to <u>screw speed</u>.

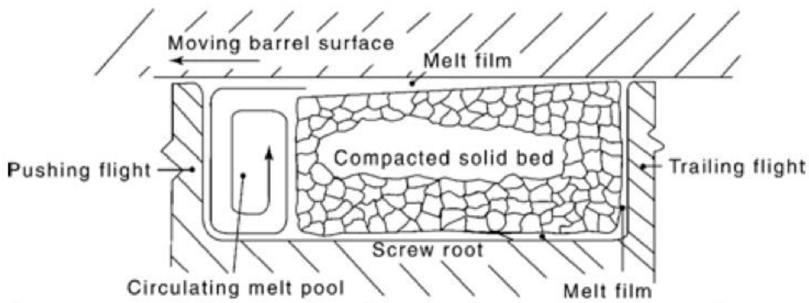
How <u>hard you press</u> is analogous to <u>backpressure</u>.

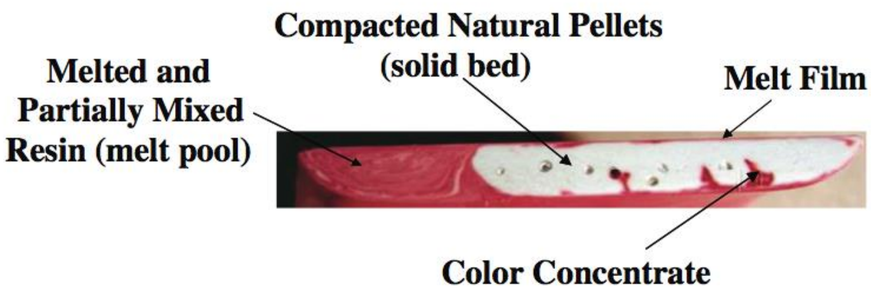
Screw Melting

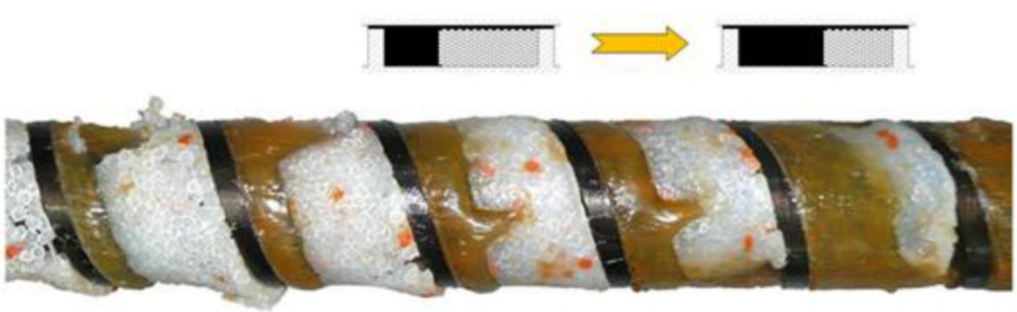
As pellets fall onto the screw they begin to heat. Most of this heat comes from the heater bands.



As material moves forward on the screw the pellets are compacted and a melt pool begins to form. The images below illustrate this melting.



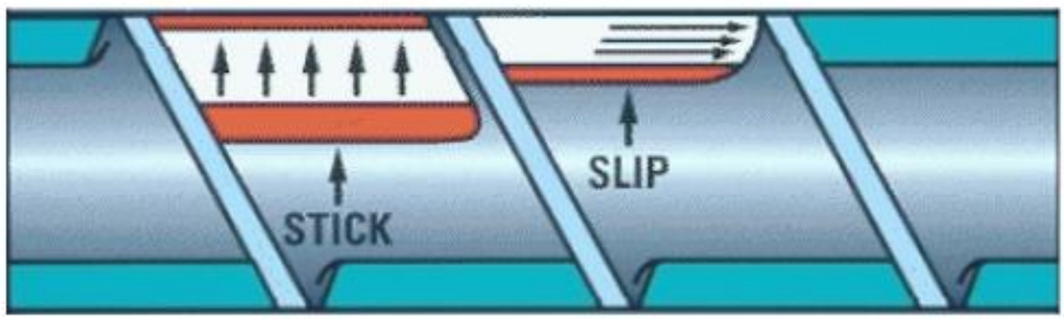




Drag, Pressure, and Leakage Flow

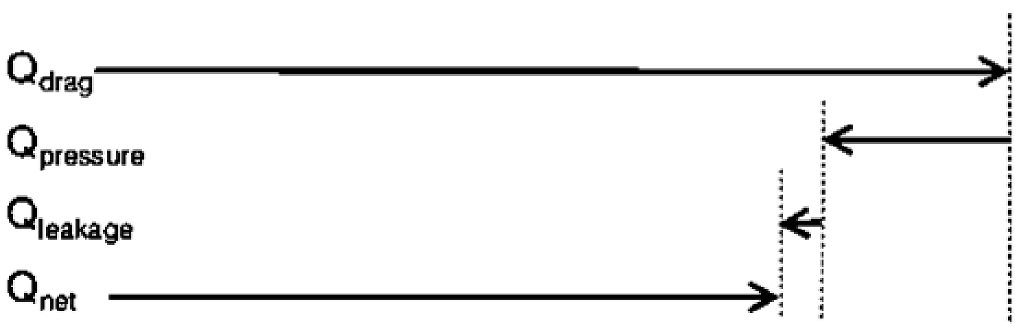
As the screw moves forward the melt is pulled forward by the action of the screw and barrel. This is called drag-flow.

For the plastic to move forward the plastic MUST stick to the barrel and slip on the screw. If this doesn't happen the screw doesn't recover!



Pressure flow and leakage flow both work against drag flow to reduce the volumetric output of the screw.

$$Q_{net} = Q_{drag} - Q_{Pressure} - Q_{leakage}$$



Heat Input

The profile chosen will impact how much heat is provided by the screw (through shear heating) and by the heat bands.

General Rule of Thumb:

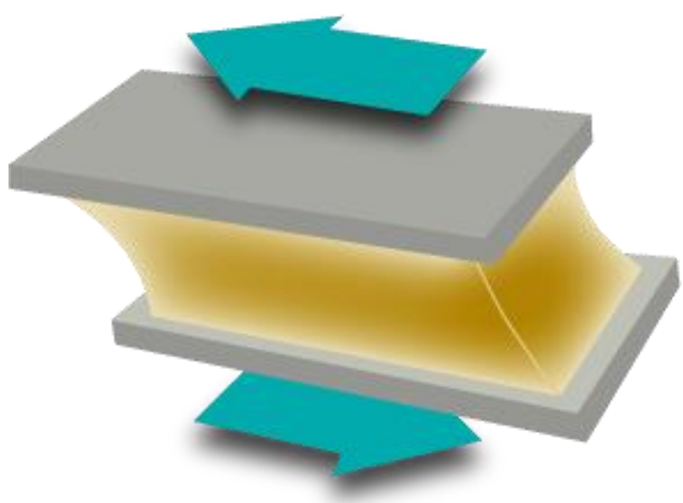
As much heat should be provided by shear as is reasonable for any material.

The amount of shear input to an injection molding material is based on two factors:

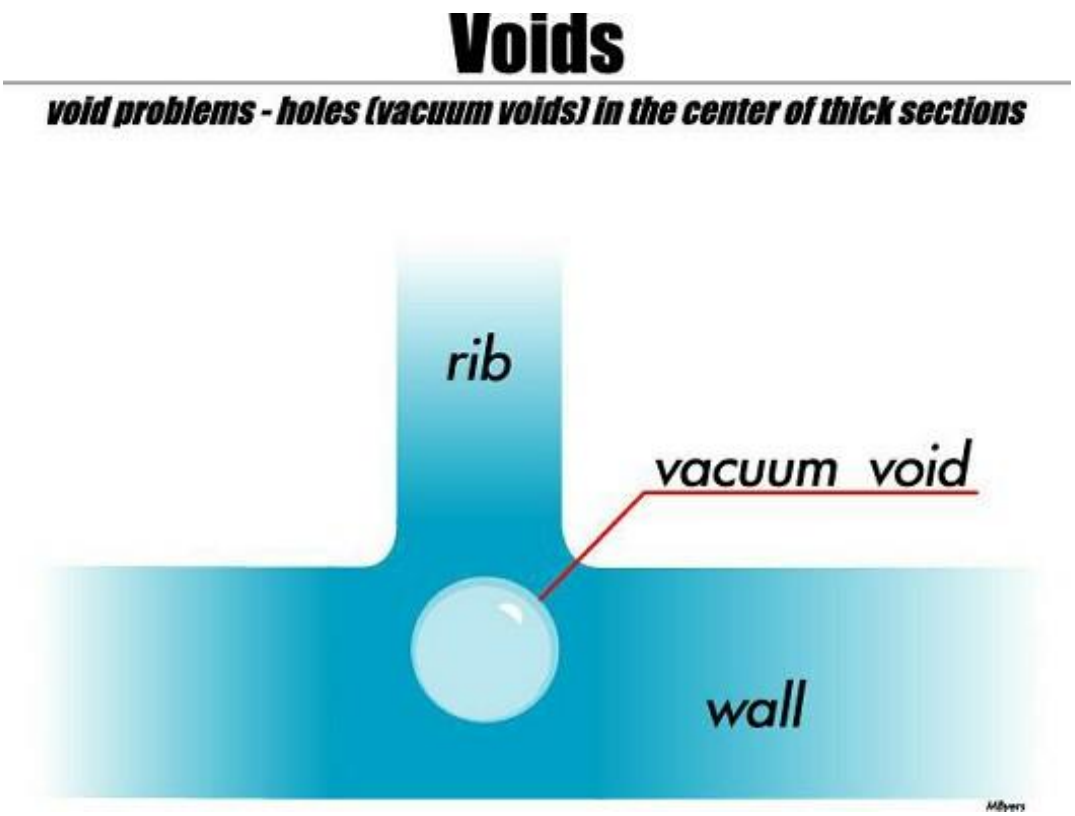
- Speed (provided by screw speed)
- Pressure (provided by backpressure)

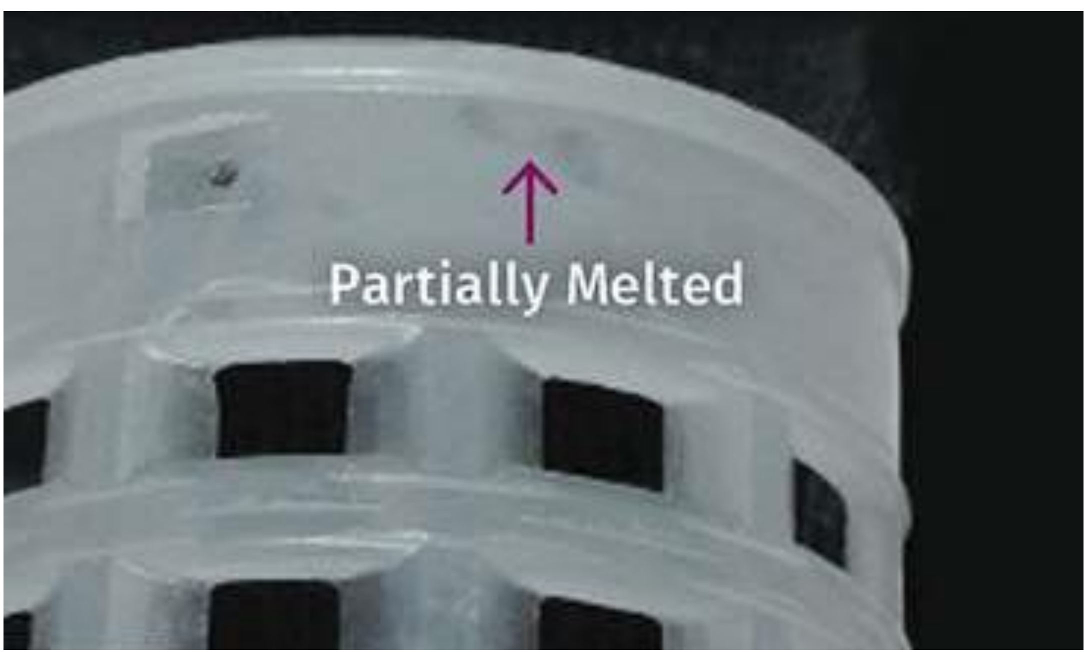
These two factors not only determine the amount of shear heating they also determine the amount of mixing done on the screw.

- Mixing color concentrate and natural resin
- Mixing back regrind
- Mixing blends



Air bubbles in your parts? Not likely.



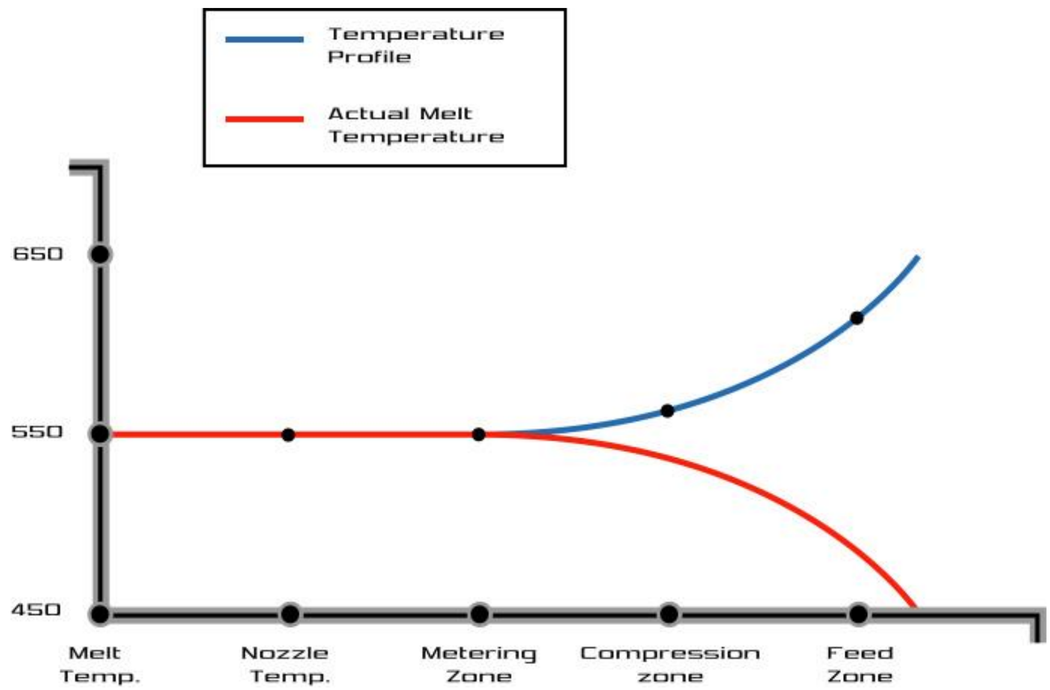


Setting the Right Temperature Profile

Selecting the Correct Melt Temperature

To select a proper set temperature profile for barrel heats we will use a multistep estimate and refine the process.

1. Estimate a melt temperature
2. Select the correct temperature profile
3. Establish the correct tacking temperature
4. Measure the actual melt temperature



Screw Recovery Delay

When developing a shot of plastic in Injection Molding the screw should recover just in time for the next Injection. If the screw sits too long the shot may begin to develop inconsistencies in both composition and thermal uniformity. If the screw recovery takes too long the cycle will be delayed. Both conditions are not desirable.

Estimate a melt temperature

There are several ways to estimate the correct melt temperature range. The overarching goal in picking this range is to assure the pellets are at a temperature above Tg before they enter the compression zone of the screw.

•Cycle Sheets

When available, cycle sheets based on prior production runs or engineering trials are the best source for melt temperature for any given mold/machine/material combination.

•Vendor Literature

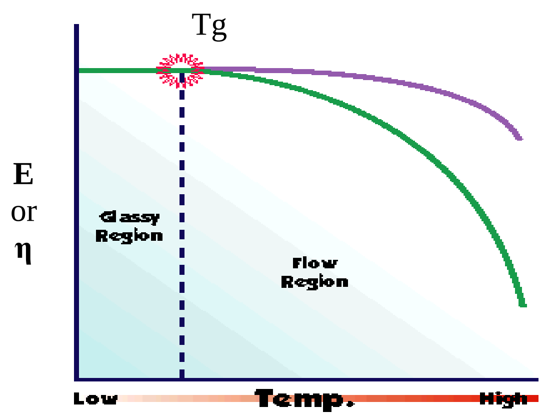
Vendor literature often can provide insights into appropriate melt temperatures for Injection Molding applications. The weakness of this data is that it is not specific for a mold/machine/material combination.

•Common "Rules of Thumb"

When prior cycle sheets and vendor literature are not available (a common occurrence in industry) two rules of thumb can be used to help determine a beginning melt temperature.

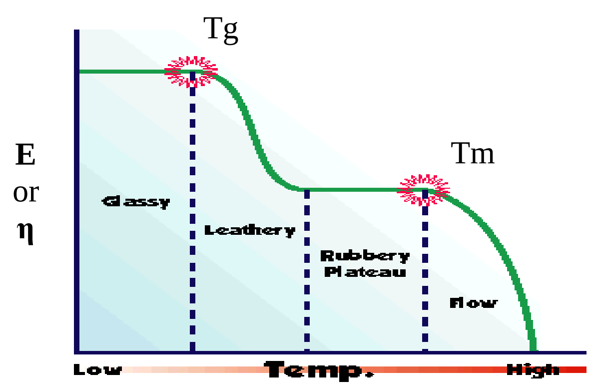
Rule # 1:
For Amorphous Materials

T_g + 150ºF



Rule # 2:
For Crystalline Materials

T_m + 50ºF



Select the correct temperature

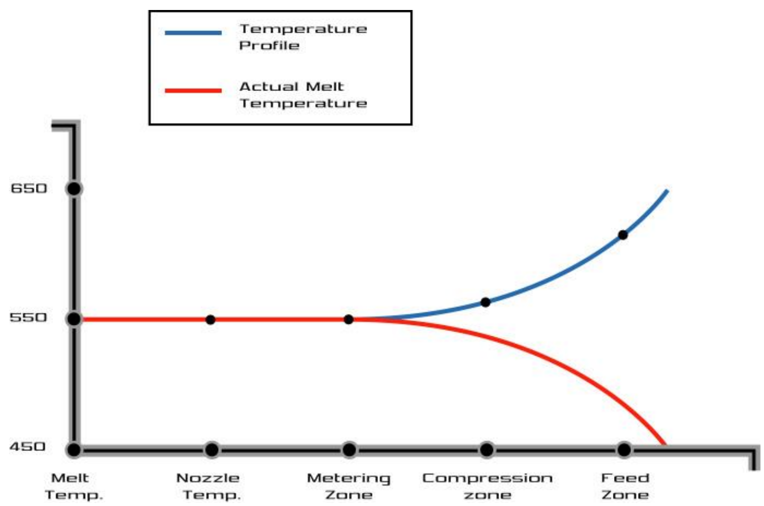
Increasing Profile

- Feed zone lower than compression zone lower than metering zone
- Use to increase shear input to the material
- Used with ductile materials

Decreasing Profile (aka: reverse profile)

- Feed zone higher than compression zone higher than metering zone
- Use to decrease shear input to the material
- Used with brittle materials

NOTE: A decreasing profile will heat pellets faster in the feed zone. It is unlikely the plastics in the feed zone will ever reach the zone set temperature. A high feed zone temperature preheats the pellets faster, so we assume the material is above Tg entering the compression zone.



Flat Profile

- All heater band zones set equal.

"Over the Mountain" Profile

- Compression Zone is set higher than Feed and Metering zones.

"Through the Valley" Profile

- Compression Zone is set lower than Feed and Metering zones.

Increasing Profile or Decreasing Profile?

Remember:

As much heat should be provided by shear as is reasonable for any material.

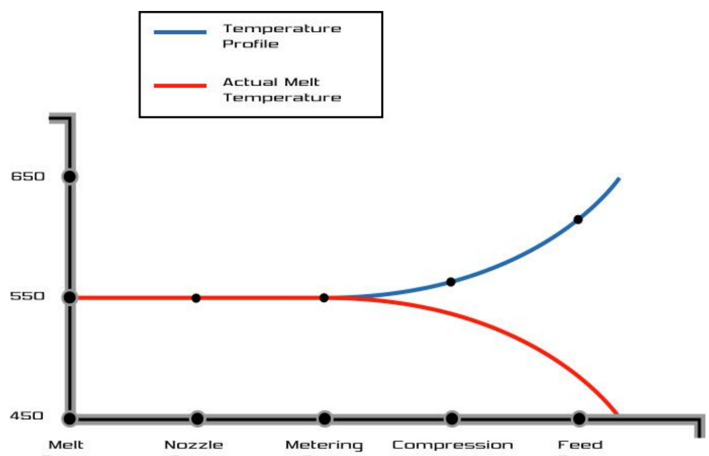
Let the screw and frictional shear heating do the work of heating the material if possible.

For ductile materials (crystalline materials where Tg > T room < Tm) reduce the feed zone temperature. Let the screw and shear do the heating.

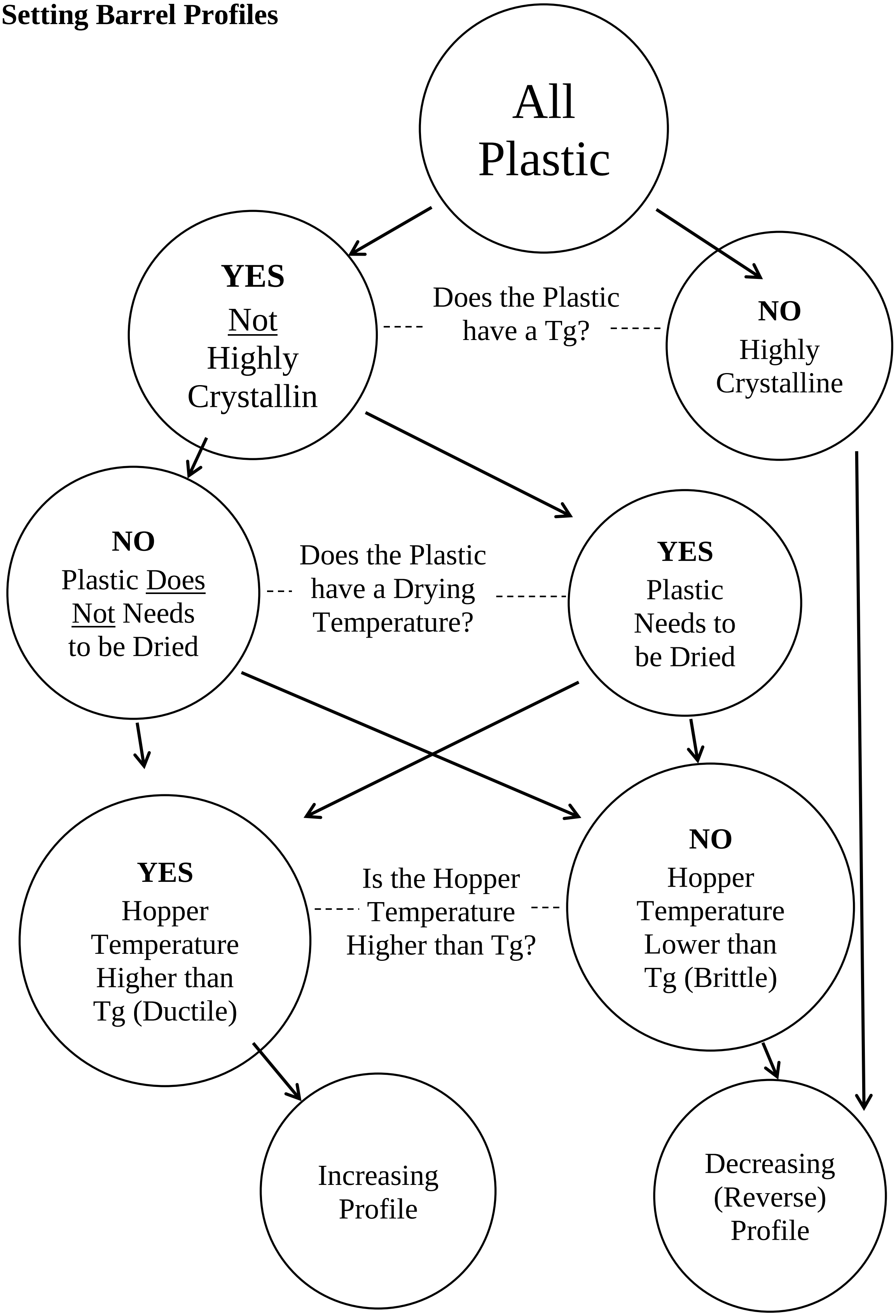
For brittle materials (amorphous and crystalline materials where Tg < T room).

The key is to assure the material is above Tg (not brittle) before pellets enter the compression zone.

In a decreasing profile, pellets fall onto the hopper at hopper temperature and begin to heat. The barrel zone heater setpoint is how much heat is being driven into the pellet, not the actual pellet temperature! Setting a feed zone temperature of 600°F does not mean the material will ever see 600°F!



Setting Barrel Profiles



Material: ______________________

What do we need to know?	Test?	Result?	Answer
MELT TEMPERATURE			
Amorphous or Crystalline?	Is there a Tm?	Yes=Crystalline No=Amorphous	
Melt Temperature?	Amorphous or Crystalline?	Amorphous = Tg+150F Crystalline = Tm + 50F	
FYI – Ductile or Brittle at Room Temperature?			
Room T Ductile or Brittle?	Is Tg greater than 72F?	Yes=RT Brittle No= RT Ductile	
What PROFILE?			
Highly Crystalline?	Is there a Tg?	Yes=Need more information, Go to Next Question No= Decreasing Profile, Enter Decreasing Profile	
Drying Temperature?	Is there a drying Temperature?	Yes=Drying T is Hopper T, Enter Drying T No=Room T is Hopper T, Enter Room T (72F)	
Hopper Ductile or Brittle?	Is Tg greater than Hopper T	Yes=HT Brittle No= HT Ductile	
Increasing or Decreasing Profile?	Is Hopper Ductile or Brittle?	Hopper T is Ductile = Increasing Profile Hopper T is Brittle = Decreasing Profile	
What rate of increase or decrease?	How far is Hopper T from Tg?	Hopper T to Tg = 0 to 50F, rate of 10F / zone Hopper T to Tg = 50 to 150F, rate of 20 F / Zone Hopper T to Tg > 150F, rate of 40F / zone	
PROFILE			
Melt T	from above		
Nozzle T	same as Melt T		
Metering Zone T	same as Melt T		
Compression Zone T	Metering Zone + Rate		
Feed Zone T	Compression + Rate		

NOTE: The profiles calculated by this method are a starting point for the tacking temperature experiment.
NOTE: Screw Speed, Backpressure and % Barrel Usage will impact the final profile.

LDPE = Low Density Polyethylene, **HDPE** = High Density Polyethylene, **P/P** = Polypropylene, **PC** = Polycarbonate, **HIPS** = High Impact Polystyrene, **PVC** = Polyvinyl Chloride,

	Glass	Crystalline	Dryer or	*Amorphous*	*Room*	*Hopper*	*Increasing*						
	Transition	Melting	*Hopper*	*or*	*Ductile*	*Ductile*	*or*						
	Temperature	Temperature	*Temperature*	*Crystalline*	*Brittle*	*Brittle*	*Decreasing*						
	Tg	Tm		A or C	D or B	D or B	I or D	Melt	Nozzle	Metering	Compression	Feed	Rate
LDPE	-202	221	72										
HDPE	-193	275	72										
P/P	-4	350	72										
HIPS	212		72										
PVC	188		72										
PMMA	221		180										
ABS	240		190										
PC	293		250										
PSU	370		300										
PPS	191	554	300										
PET	166	482	340										
PBT		500	250										
POM	-58	358	210										
PA6	122	419	180										
PA 11		383	180										
PA 12		365	180										
PA 6,6	131	486	180										
PA 6,10	122	423	180										
PEEK		649	310										
PAI	527		350										
PEI	420		300										
* aPET	167		340										
* cPET	167	491	340										

PMMA = Polymethyl methacrylate, **PA 6,6** = Nylon 6,6, **PVDC** = Polyvinylidene Chloride, **ABS** = Acrylonitrile-Butadiene-Styrene Polymer, **PPS** = Polyphenylene Sulfide, **PET** = Polyethylene Terephthalate, **PBT** = Polybutylene Terephthalate, **POM** = Polyoxymethylene (acetal), **PSU** = Polysulfone, **PEEK** = Polyehteretherketone, **PEI** = Polyehterimide, **PAI** = Polyamide-imide

Material: Polypropylene (P/P)

What do we need to know?	Test?	Result?	Answer
MELT TEMPERATURE			
Amorphous or Crystalline?	Is there a Tm?	Yes=Crystalline No=Amorphous	Yes=Crystalline
Melt Temperature?	Amorphous or Crystalline?	Amorphous = Tg+150F Crystalline = Tm + 50F	400F (Tm of 350F + 50F)
FYI – Ductile or Brittle at Room Temperature?			
Room T Ductile or Brittle?	Is Tg greater than 72F?	Yes=RT Brittle No= RT Ductile	Room T Ductile (Tg of -4F < 72F)
What PROFILE?			
Highly Crystalline?	Is there a Tg?	Yes=Need more information, Go to Next Question No= Decreasing Profile, Enter Decreasing Profile	Yes = Next Question
Drying Temperature?	Is there a drying Temperature?	Yes=Drying T is Hopper T, Enter Drying T No=Room T is Hopper T, Enter Room T (72F)	Hopper T = 72F
Hopper Ductile or Brittle?	Is Tg greater than Hopper T	Yes=HT Brittle No= HT Ductile	Hopper T Ductile (Tg of -4F < 72F)
Increasing or Decreasing Profile?	Is Hopper Ductile or Brittle?	Hopper T is Ductile = Increasing Profile Hopper T is Brittle = Decreasing Profile	Increasing Profile
What rate of increase or decrease?	How far is Hopper T from Tg?	Hopper T to Tg = 0 to 50F, rate of 10F / zone Hopper T to Tg = 50 to 150F, rate of 20 F / Zone Hopper T to Tg > 150F, rate of 40F / zone	Rate = 20F / Zone (Hopper T to Tg = 76F)
PROFILE			
Melt T	from above		400F
Nozzle T	same as Melt T		400F
Metering Zone T	same as Melt T		400F
Compression Zone T	Metering Zone + Rate	Increasing Rate = 20F	380F
Feed Zone T	Compression + Rate	Increasing Rate = 20F	360F

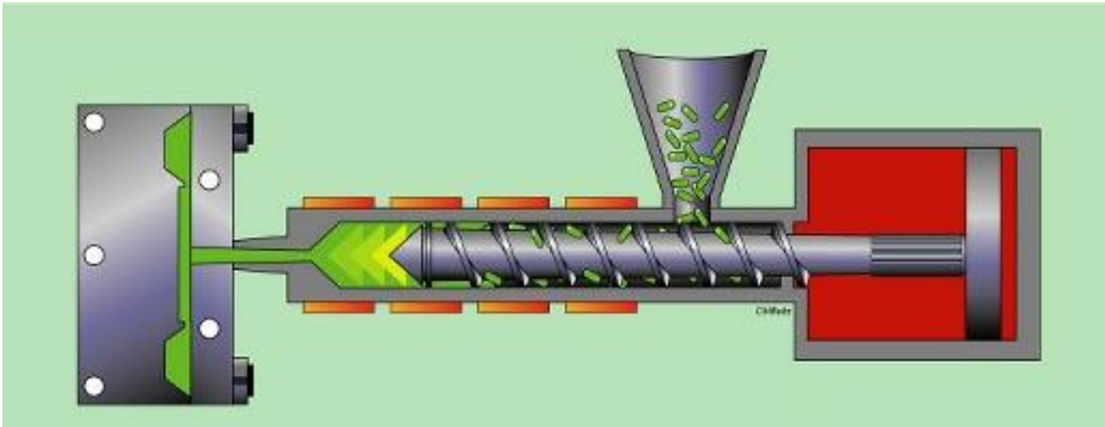
NOTE: The profiles calculated by this method are a starting point for the tacking temperature experiment.
NOTE: Screw Speed, Backpressure and % Barrel Usage will impact the final profile.

Material: Polystyrene (P/S)

What do we need to know?	Test?	Result?	Answer
MELT TEMPERATURE			
Amorphous or Crystalline?	Is there a Tm?	Yes=Crystalline No=Amorphous	No=Amorphous
Melt Temperature?	Amorphous or Crystalline?	Amorphous = Tg+150F Crystalline = Tm + 50F	362F (Tg of 212F + 150F)
FYI – Ductile or Brittle at Room Temperature?			
Room T Ductile or Brittle?	Is Tg greater than 72F?	Yes=RT Brittle No= RT Ductile	Room T Brittle (Tg of 212F > 72F)
What PROFILE?			
Highly Crystalline?	Is there a Tg?	Yes=Need more information, Go to Next Question No= Decreasing Profile, Enter Decreasing Profile	Yes = Next Question
Drying Temperature?	Is there a drying Temperature?	Yes=Drying T is Hopper T, Enter Drying T No=Room T is Hopper T, Enter Room T (72F)	Hopper T = 72F
Hopper Ductile or Brittle?	Is Tg greater than Hopper T	Yes=HT Brittle No= HT Ductile	Hopper T Brittle (Tg of 212F > 72F)
Increasing or Decreasing Profile?	Is Hopper Ductile or Brittle?	Hopper T is Ductile = Increasing Profile Hopper T is Brittle = Decreasing Profile	Decreasing Profile
What rate of increase or decrease?	How far is Hopper T from Tg?	Hopper T to Tg = 0 to 50F, rate of 10F / zone Hopper T to Tg = 50 to 150F, rate of 20 F / Zone Hopper T to Tg > 150F, rate of 40F / zone	Rate = 20F / Zone (Hopper T to Tg = 140F)
PROFILE			
Melt T	from above		362F
Nozzle T	same as Melt T		362F
Metering Zone T	same as Melt T		362F
Compression Zone T	Metering Zone + Rate	Decreasing Rate = 20F	382F
Feed Zone T	Compression + Rate	Decreasing Rate = 20F	402F

NOTE: The profiles calculated by this method are a starting point for the tacking temperature experiment.
NOTE: Screw Speed, Backpressure and % Barrel Usage will impact the final profile.

% Barrel Usage / Heat History



Barrel Usage

% Barrel Used = (Shot Size-Cushion) (in.) / Maximum Shot Capacity (in.)) / 100

To produce proper mixing and shear on the screw 25% - 65% of the screw should be turned over on every shot.

Heat History

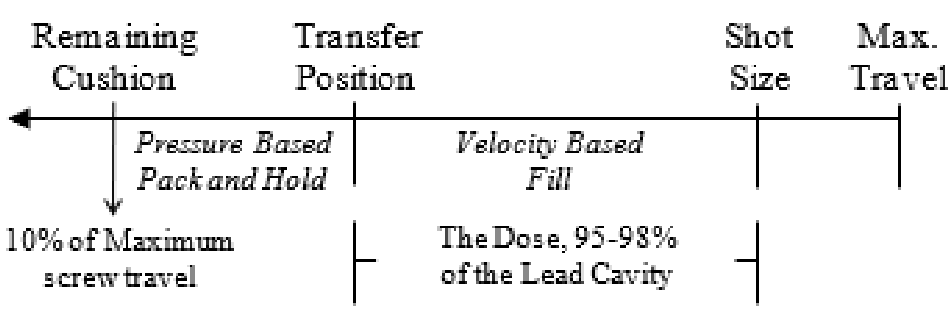
Shots on the Screw = (Maximum Shot Capacity (in.) / ((Shot Size-Cushion) (in.))

The more shots that sit on the screw the more thermal degradation of the plastic.

Residence Time

Screw Residence Time = Shots on the Screw x Cycle Time

The amount of thermal degradation will also be impacted by cycle time.



Establish the correct tacking temperature

The tacking temperature for any material is determined by observing the minimum time necessary for screw recovery.

1. Pick a Screw RPM and Backpressure appropriate for the material (high shear or low shear for example).
2. Find the recommended feed zone temperature in the table above.
3. Set the initial feed zone temperature 100°F below this temperature.
4. Increase the temperature in 20°F increments.
5. Measure the screw recovery time corresponding to each feed zone temperature.
6. The correct feed zone temperature will correspond to the minimum recovery time.

Screw RPM	Backpressure (PSI)	Feed Zone Temperature (F)	Screw Recovery Time (sec)

The 30/30/30 melt temperature method

Getting an accurate measure of the melt temperature is challenging.

One popular method for getting a melt temperature measurement is the 30/30/30 method.

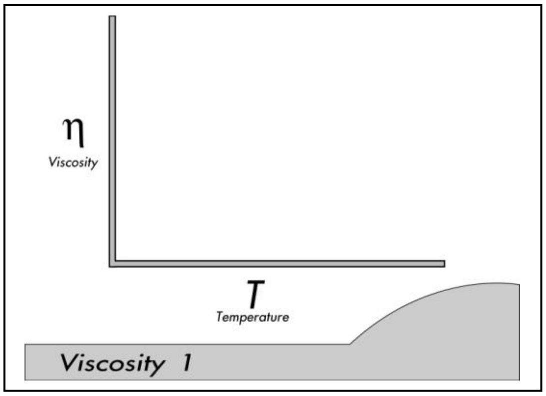
THE FIRST '30' - Let the machine stabilize for 30 shots or run for 30 minutes. This will give you a production melt temperature.

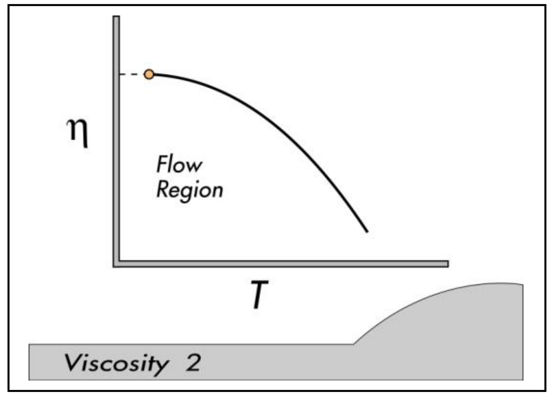
THE SECOND '30' - Preheat the pyrometer's needle to within 30°F of what you believe the melt temperature to be. This eliminates the thermal shock of a cold needle into a hot melt that will produce the insulating layer mentioned previously.

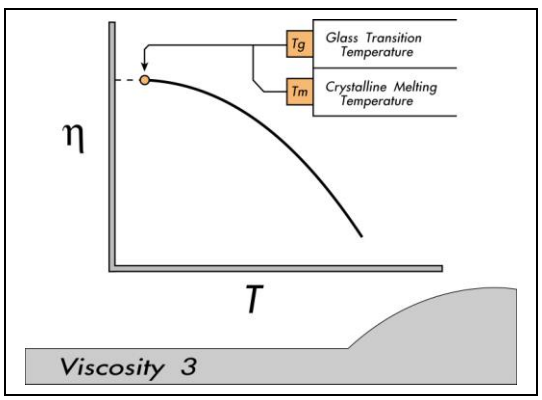
THE THIRD '30' - Put the pyrometer's needle into the center of the purge and DON'T MOVE or WIGGLE IT. Take your reading 30 seconds later.

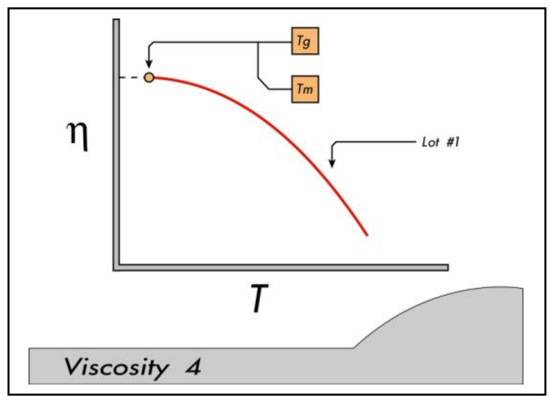
Viscosity vs. Temperature

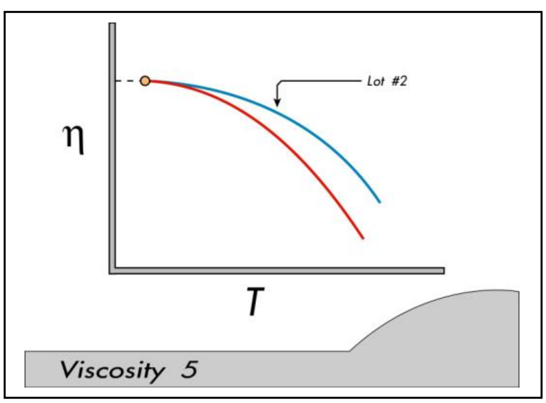
Once we have determined the correct <u>melt temperature range</u> the <u>final melt temperature</u> can be used to adjust the material melt viscosity. The following "Almost True" Story illustrates this point.

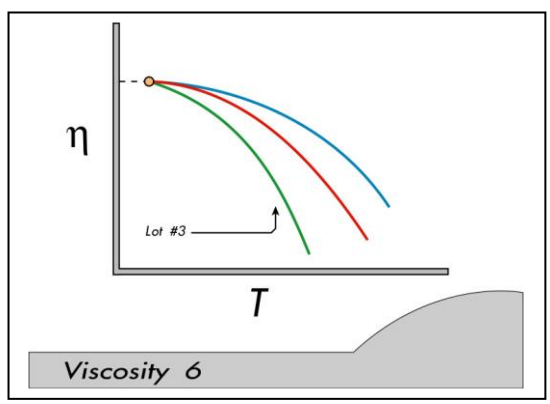


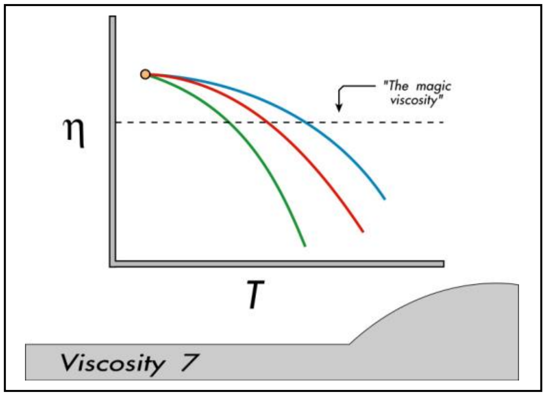




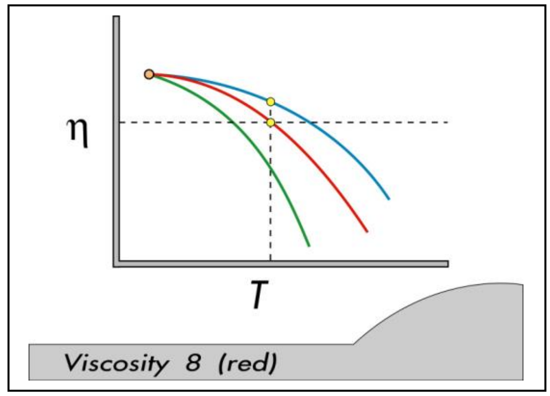




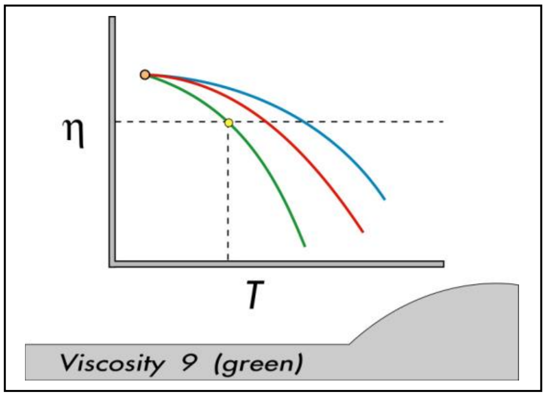




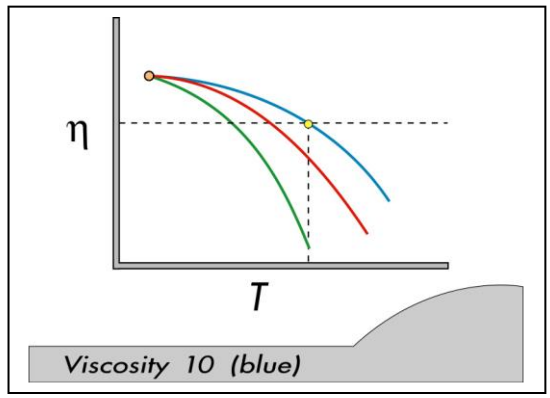
"The magic viscosity"
η
T
Viscosity 7



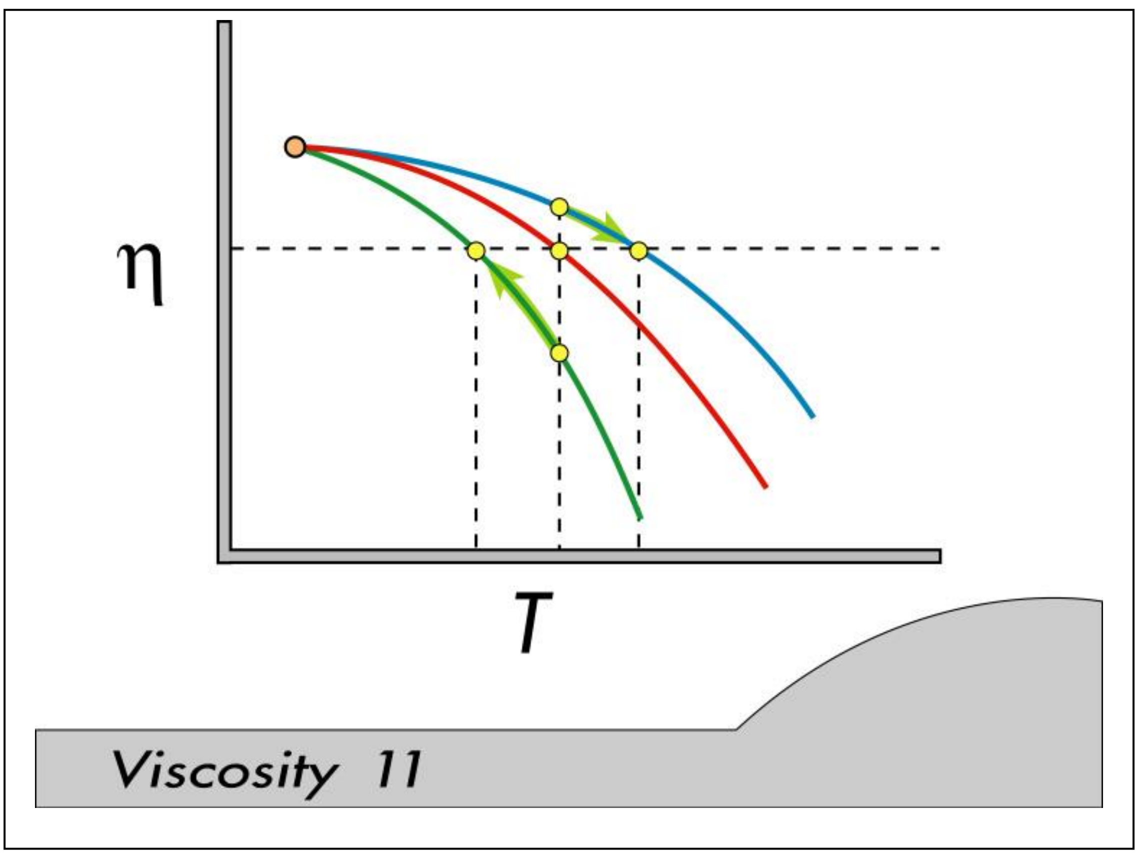
η
T
Viscosity 8 (red)



η
T
Viscosity 9 (green)

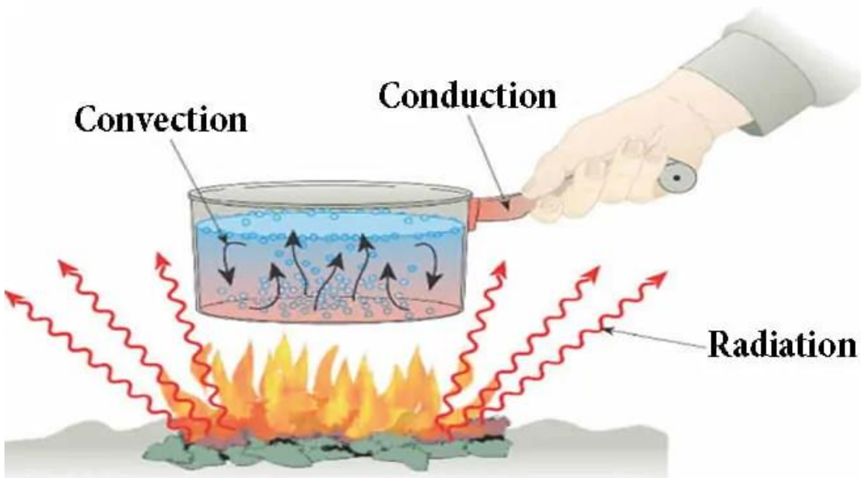


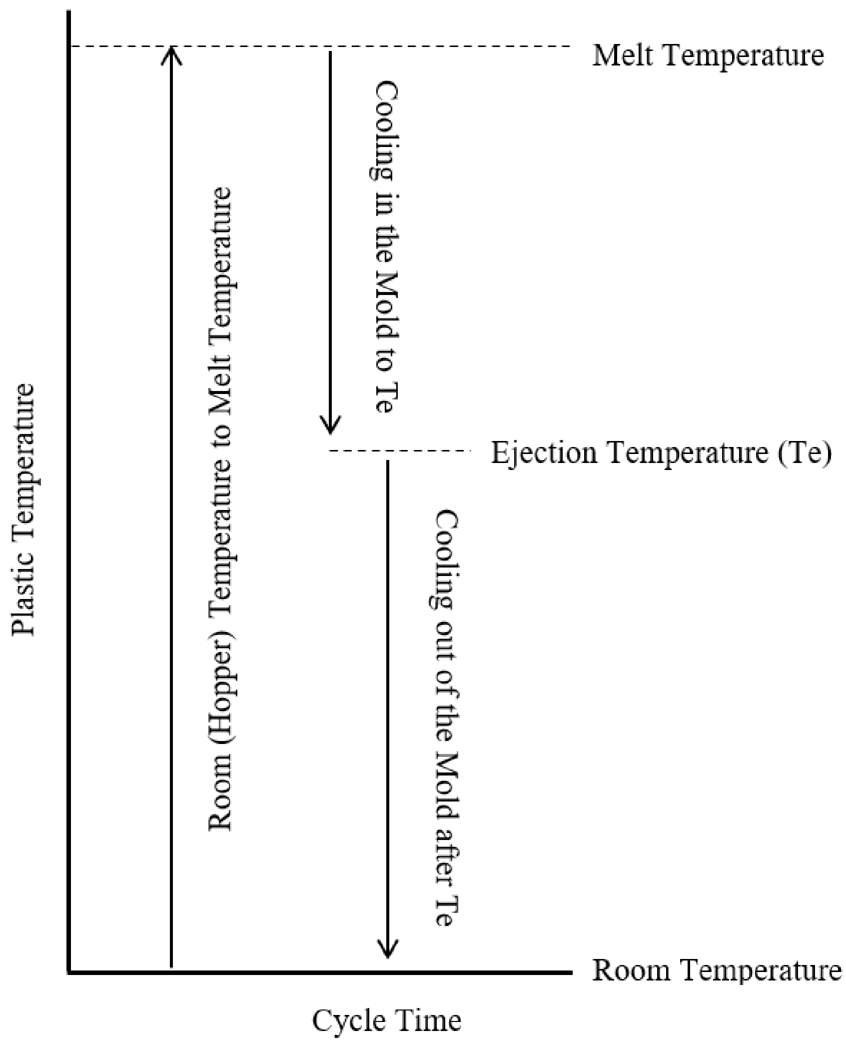
η
T
Viscosity 10 (blue)



η
T
Viscosity 11

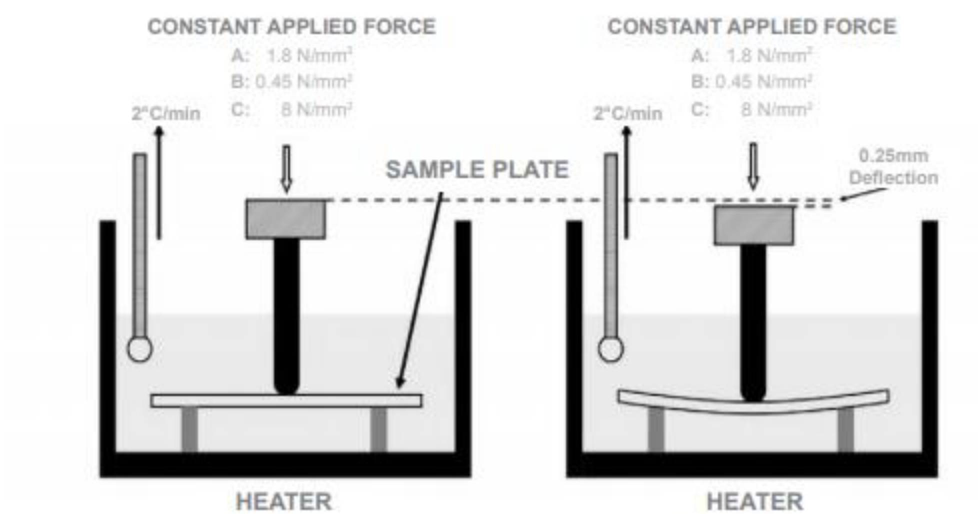
Cooling In and Out of the Mold: Delta T, Ejection Temperature



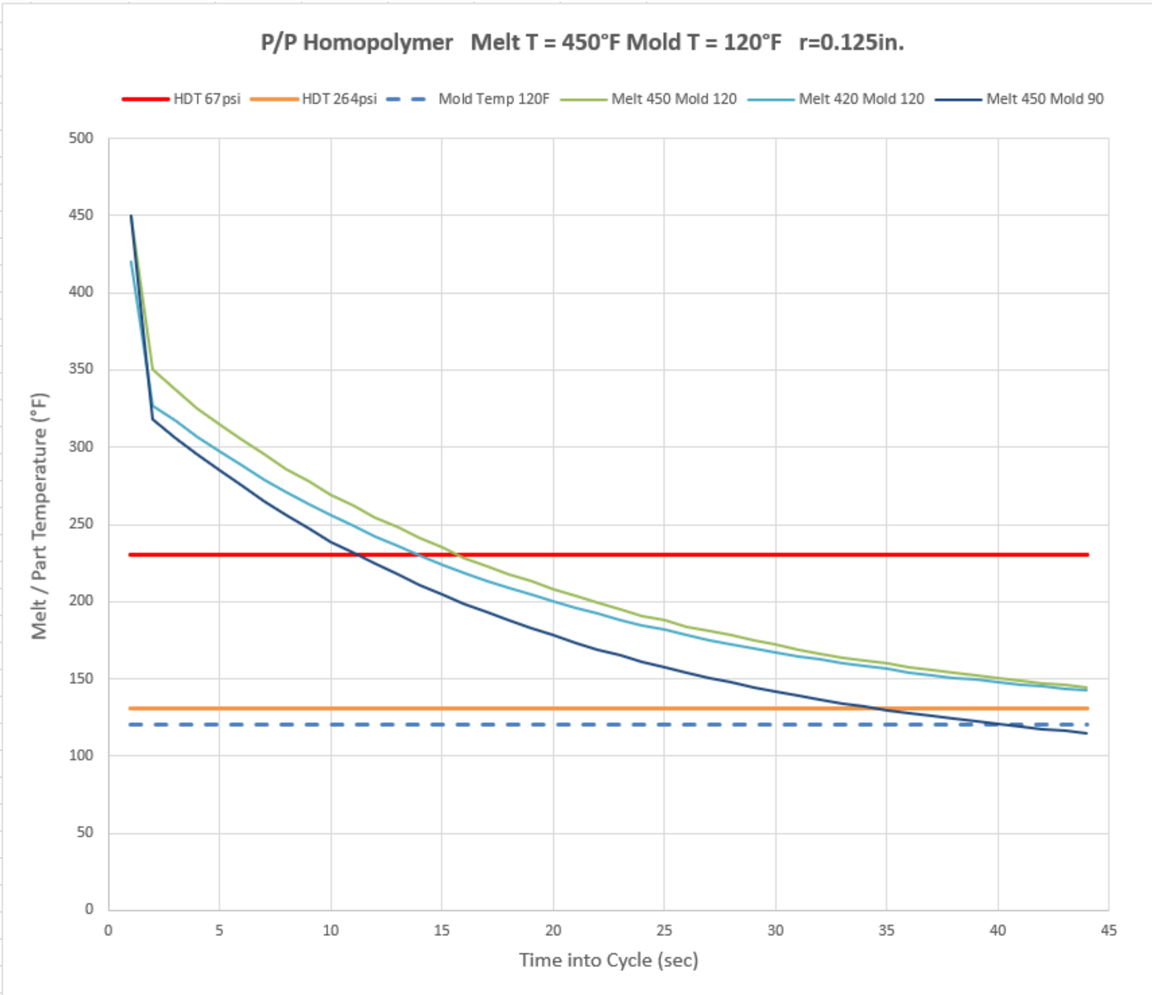


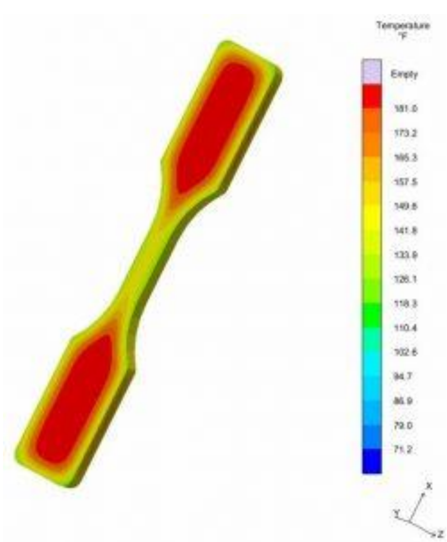
HDT Refence Temperatures

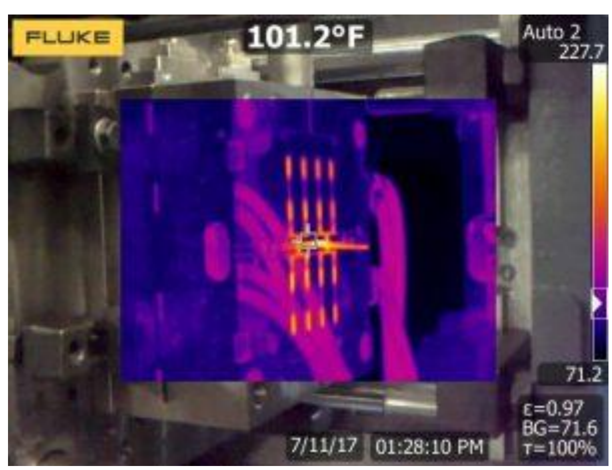
Polymer Type	Deflection Temperature at 0.46 MPa (°C)	Deflection Temperature at 1.8 MPa (°C)	Melting Point (°C)
ABS	98	88	-
ABS + 30% Glass Fiber	150	145	-
Acetal Copolymer	160	110	200
Acetal Copolymer + 30% Glass Fiber	200	190	200
Acrylic	95	85	130
Nylon 6	160	60	220
Nylon 6 + 30% Glass Fiber	220	200	220
Polycarbonate	140	130	-
Polyethylene, HDPE	85	60	130
Polyethylene Terephthalate (PET)	70	65	250
PET + 30% Glass Fiber	250	230	250
Polypropylene	100	70	160
Polypropylene + 30% Glass Fiber	170	160	170
Polystyrene	95	85	-



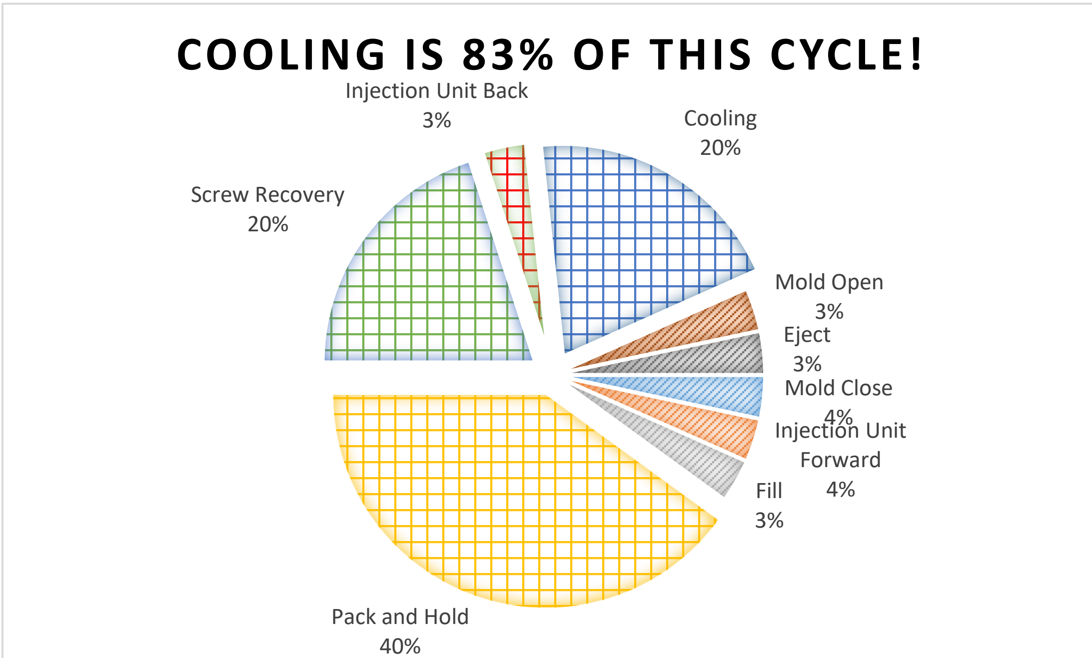
Cooling Time

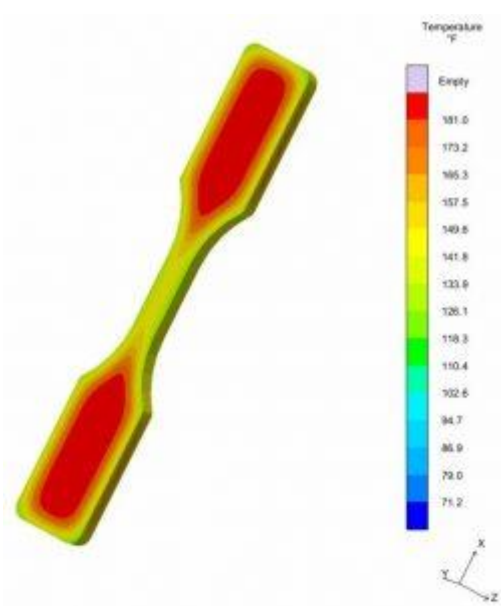


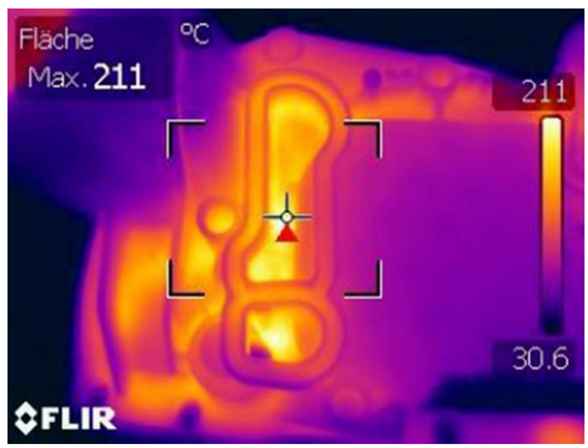


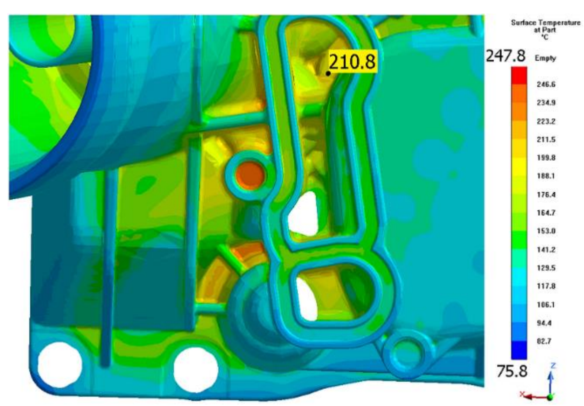


“Cooling is the Longest Part of an Injection Molding Cycle”







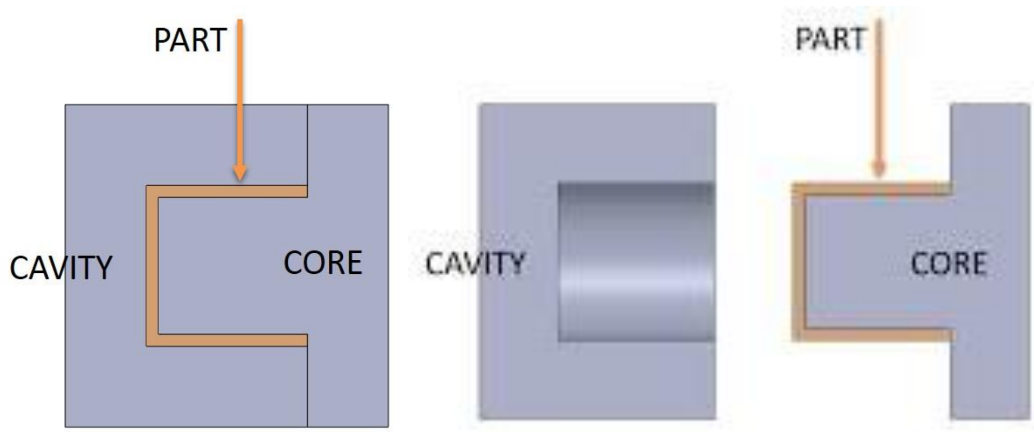


Cooling In the Mold - Moving BTUs from the Melt to the Steel

Moving BTUs

Melt → Steel → Coolant / “Drain”

If there is Melt to Steel Contact:



Conduction:

$$\mathbf{H = K\,A\,t\,(T_2 - T_1) / L}$$

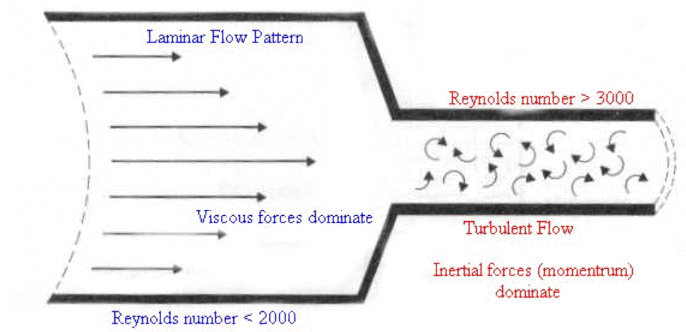
H = Heat Transferred by Conduction
K = **Plastic or Cavity Steel** material heat transfer constant
A = area of the cavity in contact with the molding material
t = beginning of fill to cooling time
T_2 = Temperature at the Part / Steel Interface
T_1 = Temperature at the Part / Steel Interface
L = Distance from the Part Center to the Part / Steel Interface

Values of *K* for different materials (BTU/°F/ft²/hr)			
Metals		Other Materials	
Stainless Steel	10	Polystyrene	0.07
Tool Steel (H-13)	12	Polypropylene	0.07
Tool Steel (P-20)	21	Nylon	0.14
Beryllium Copper	62	Polyethylene	0.18
Kirksite	62		
Brass (60-40)	70	Air	0.14
Aluminum	100	Water	0.39
Copper (pure)	222		

Cooling In the Mold - Moving BTUs from the Steel to the Coolant

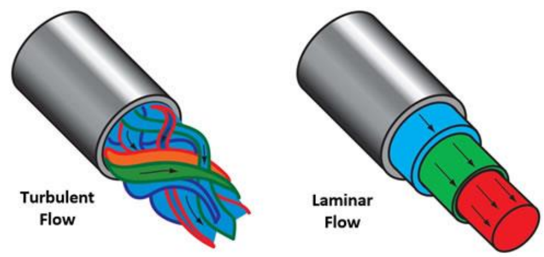
Convection

Convective cooling is cooling heat transport to a moving fluid. The efficiency of the heat transfer to a moving fluid (the circulating medium) plays a large role in determining the efficiency of mold cooling.



Laminar and Turbulent Flow - Reynolds Number

A **Reynolds Number** below 2,300 indicates Laminar Flow will occur; above 3,400 Turbulent Flow will occur, between 2,300 and 3,400 is a Transition area where Laminar Flow is in the process of breaking down to Turbulent Flow.



$$\mathbf{R = 3{,}160\ Q/d\ \eta \quad or \quad R=7{,}740\ V\ d/\ \eta}$$

where:

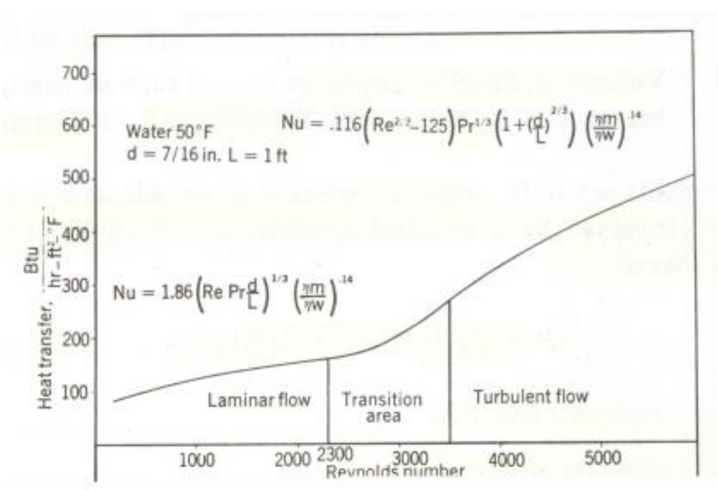
R = Reynolds Number

Q = Volumetric Flow Rate (gpm)

V = Coolant Velocity (ft/s)

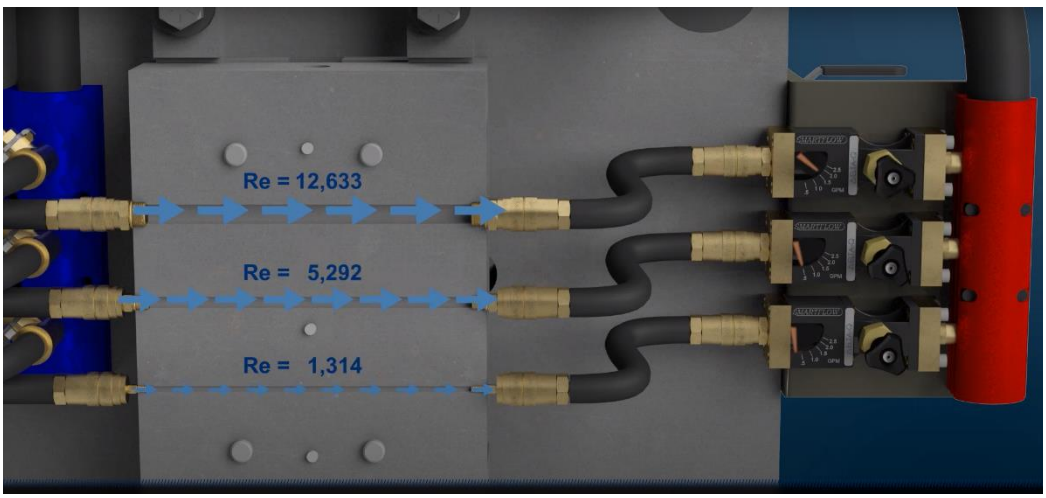
D = Diameter of flow channel (in)

η = the kinematic viscosity of water



https://www.youtube.com/watch?v=s2lpd7leGPQ

https://www.youtube.com/watch?v=KY-nDGUX7yY



https://www.youtube.com/watch?v=UvfhNoc3SC0

Moving BTUs <u>Efficiently</u> through the coolant to the “Drain”

Melt → Steel → Coolant / “Drain”

Wittmann
301 Series Flow Meter

Mold-Masters
Water Flow Monitoring
(WFM)

Smart Flow

Temperature Control Units (TCUs), Circulators, and Chillers

The water entering a mold can be supplied from a number of sources. The most common coolant delivery systems include a method of controlling the flow rate, a method for controlling the coolant temperature, and frequently, a method for removing heat from the coolant. The heat removal is typically accomplished using either a small chiller system or a closed plant evaporative system.

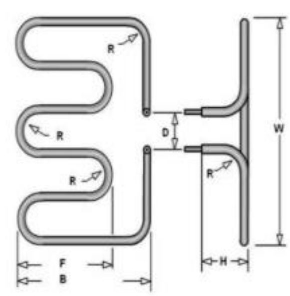
Other Cooling Media

Water is limited as a coolant by its boiling point. Many circulators claim a maximum temperature of 250°F. The pressure in the system allows operation above 212°F, the boiling point of water. At temperatures above 212°F a leak in the system allows the escaping water to turn to steam. Even at temperatures above 180°F water escaping from a leak can cause instant burns. Also, many water hoses are not rated for temperatures above 180°F. The solution for many molders is hot oil or Calrod Heaters.

Hot Oil

Hot oil is circulated through the mold in a very similar way as circulating water. Extreme caution should be taken when working with hot oil. IT IS HOT!

Calrod Heaters



Calrod Heaters for cooling? Heat the mold to 425°F to cool it? You can cook a Pizza at that temperature!!! If the melt temperature is 670°F, then 425°F is 250°F cooler! A 250°F difference between melt and mold temperature is huge when molding thin-walled parts! It may be tough to fill the part!

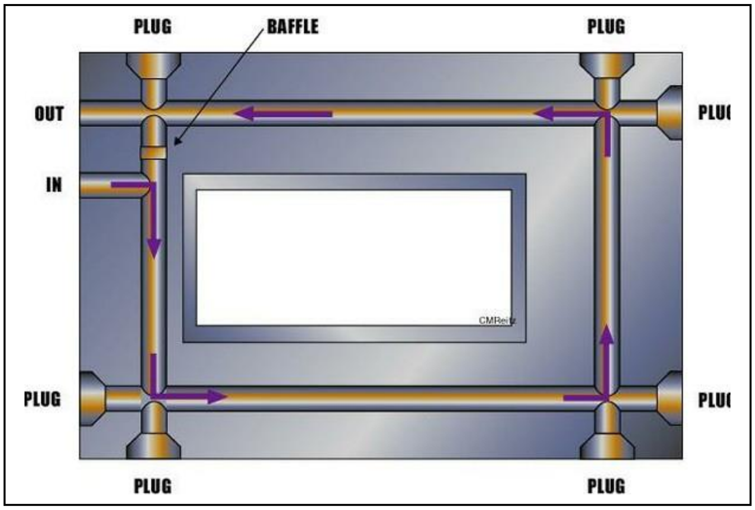
Moving BTUs <u>Equally</u> through the coolant to the "Drain"

Plate Cooling

Many molds are cooled by a process called "plate cooling".

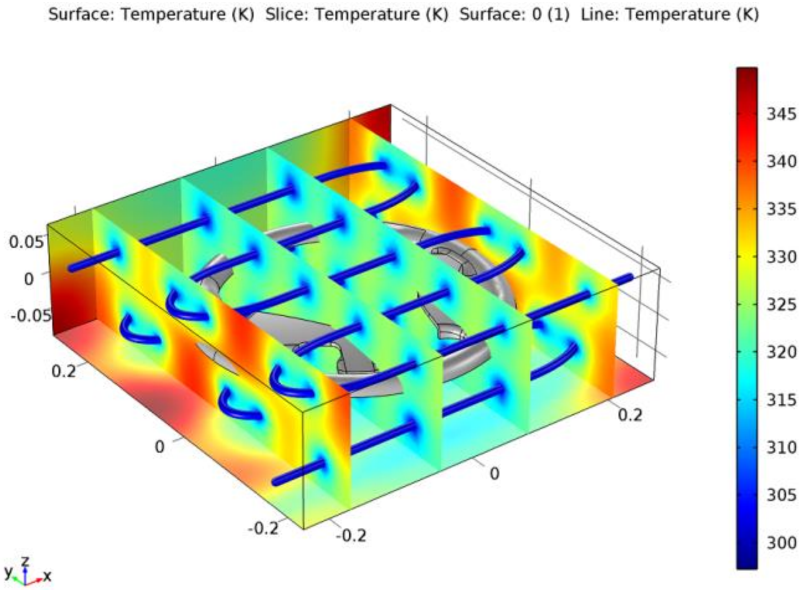
Does this type of cooling provide equal cooling to all areas of the cavity?

What about the mold shown below? Can you list three sources of unequal cooling in this system?

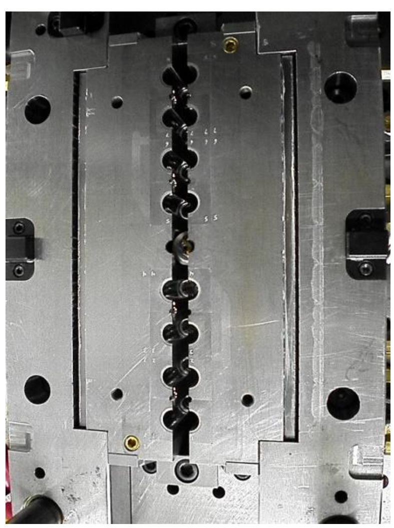


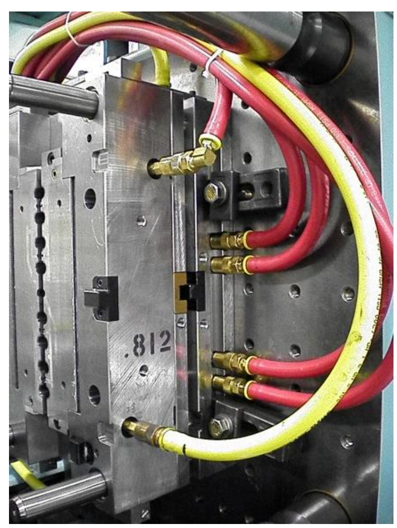
The three:
Geometry (centering), Cooling Media ΔT, and Corner / Side Issues.

Does this type of cooling provide equal cooling to all areas of the cavity?



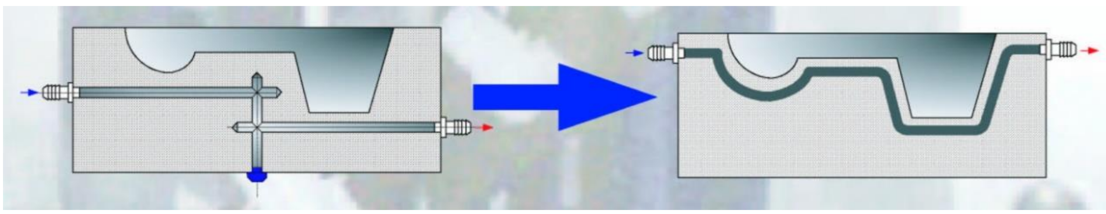
Watering the Mold, Cooling Balance

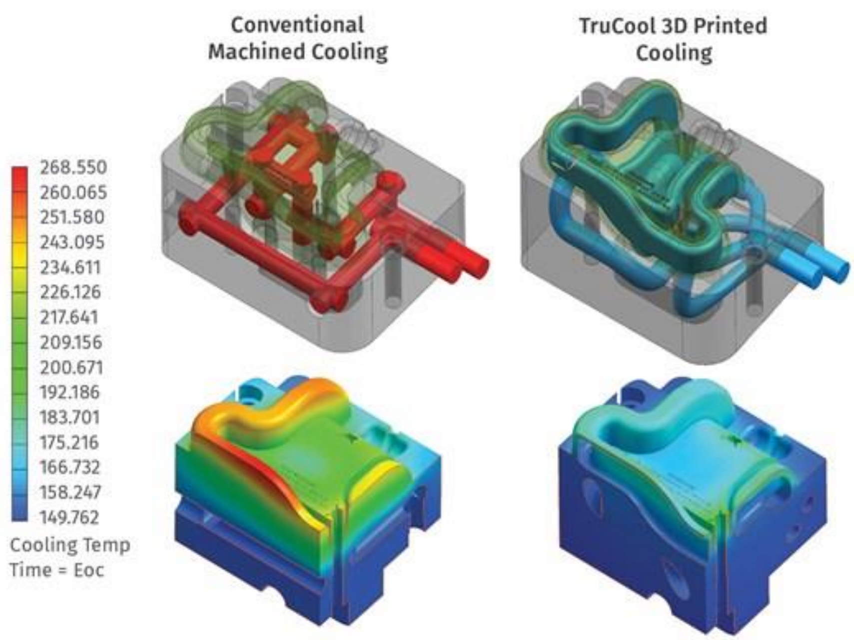




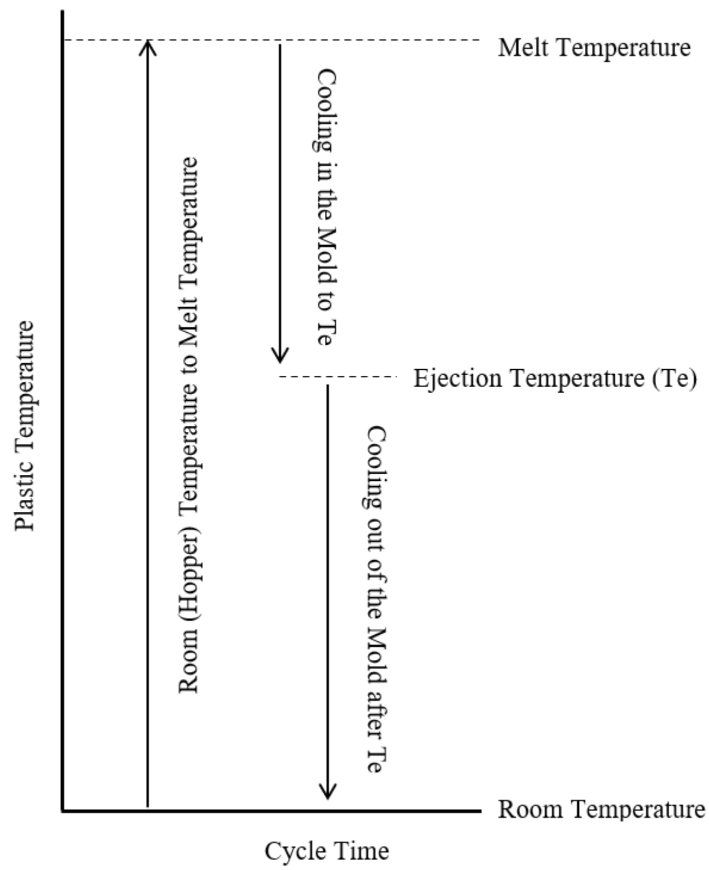
Conformal Cooling

One solution to the impossible job of cooling all surfaces equally is conformal cooling. In conformal cooling, the coolant channels are machines (or printed) to match the geometry of the part!



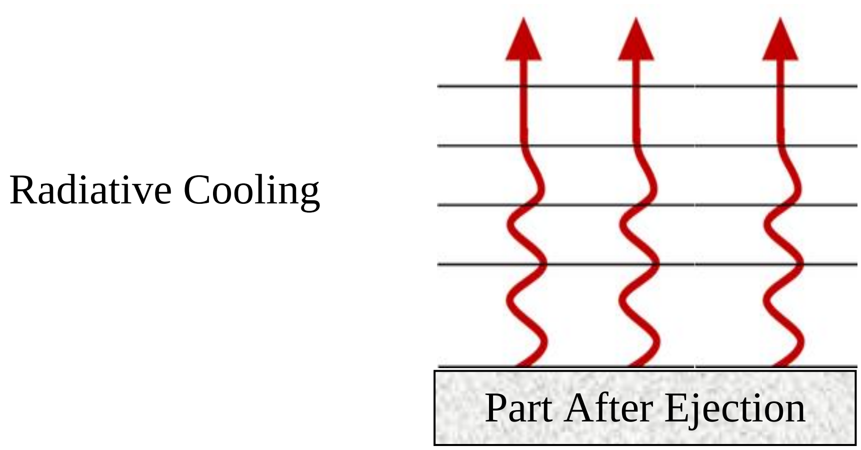


Cooling Out of the Mold (Te)



Radiative Cooling

Radiative cooling is the process by which a body loses heat by thermal radiation. As Planck's law describes, every physical body spontaneously and continuously emits electromagnetic radiation.

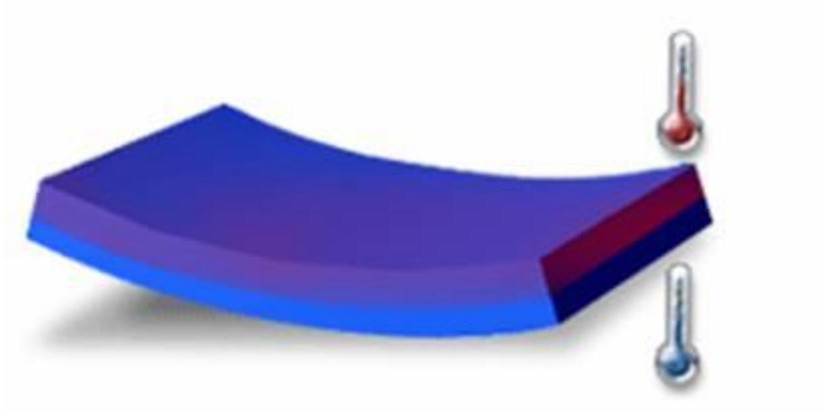


Part Defects as a result of Improper Cooling

Warp

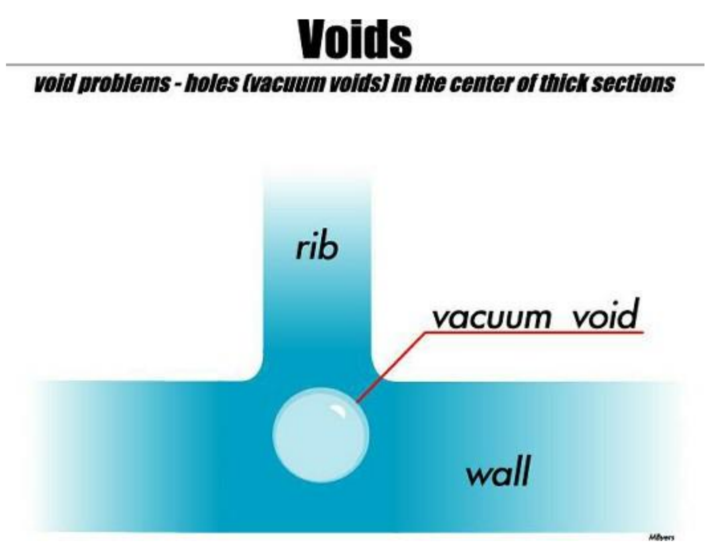
Cause: Unequal cooling.

Solution: Use water on each half to balance.

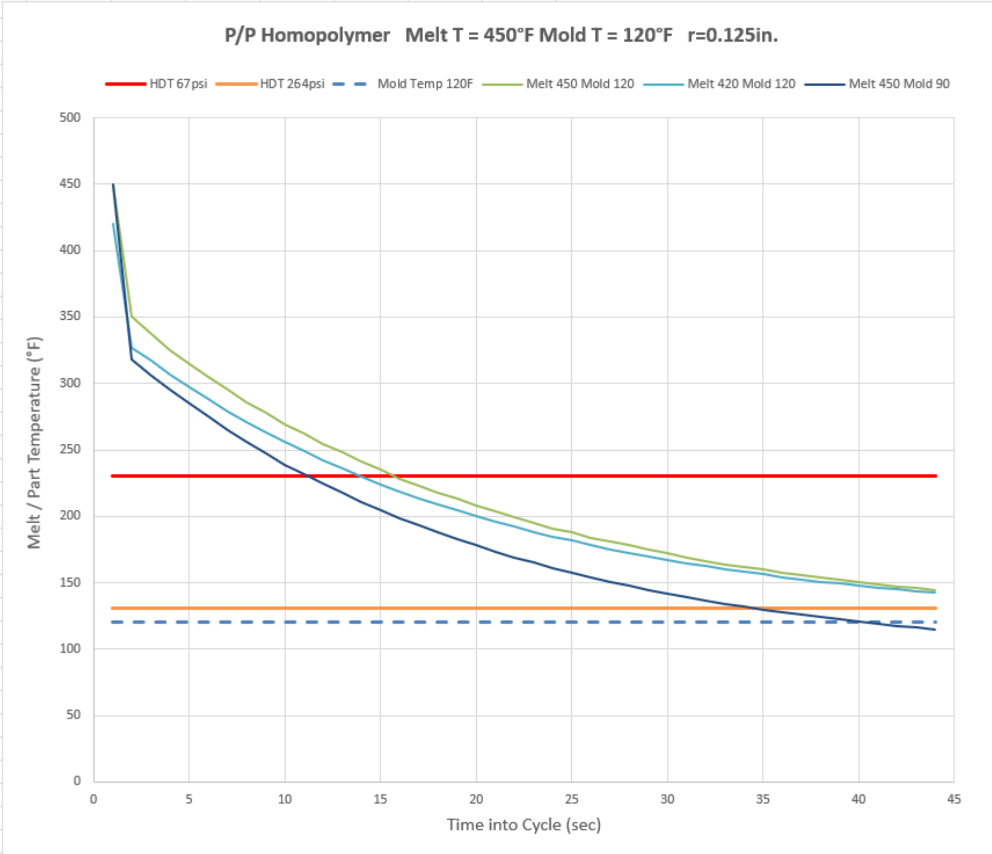


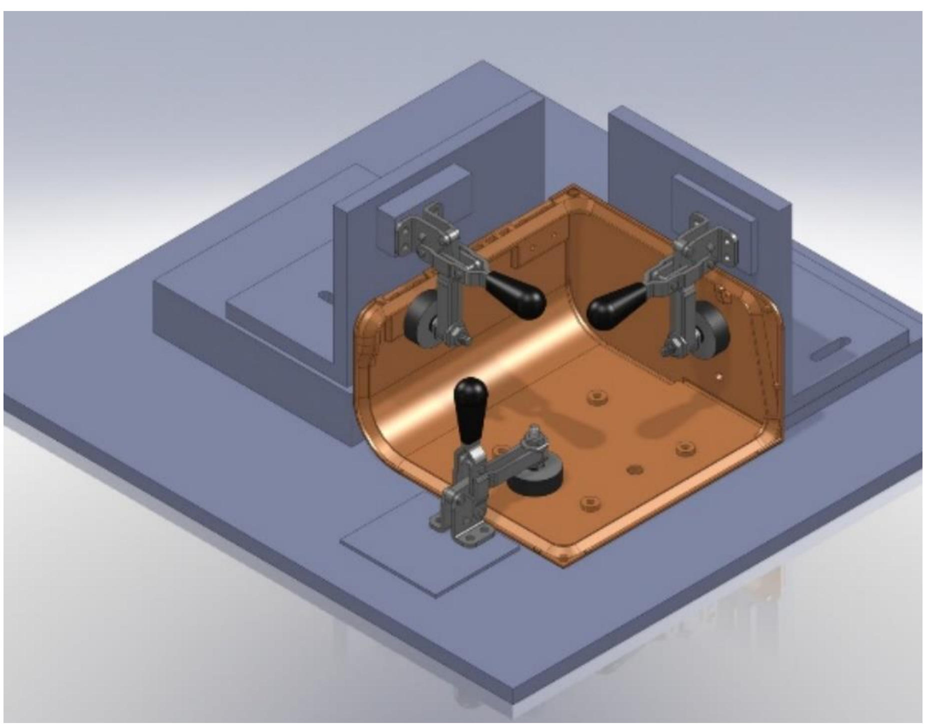
Sinks, Voids, (Residual Internal Stress)

All three defects have the same root cause, excess residual internal packing stress.

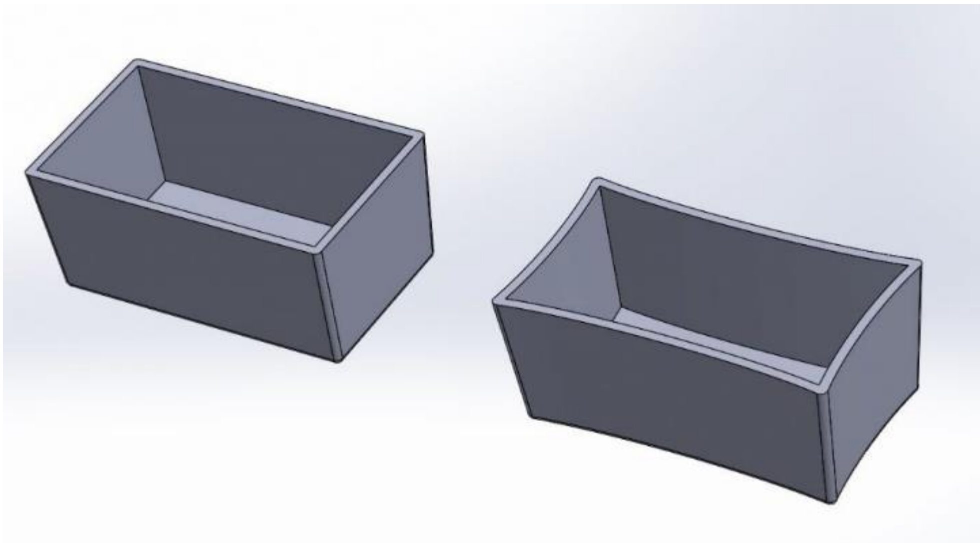


Shrink Fixturing





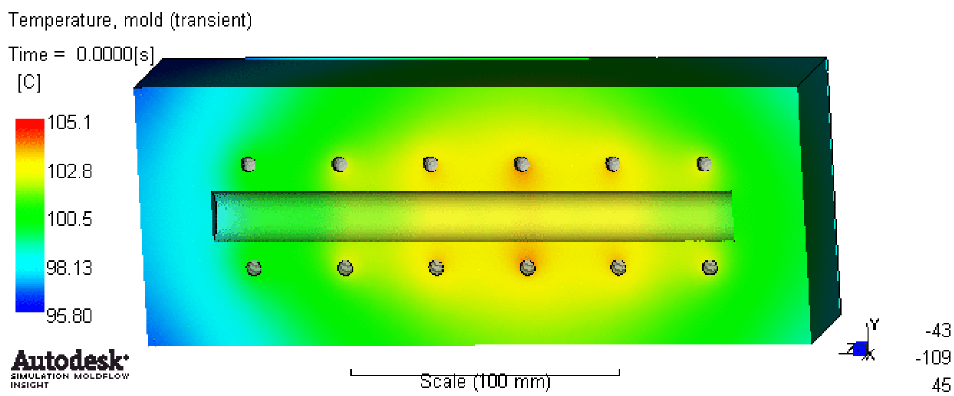
Build a Shrink Fixture or Use the Mold as a Shrink Fixture?

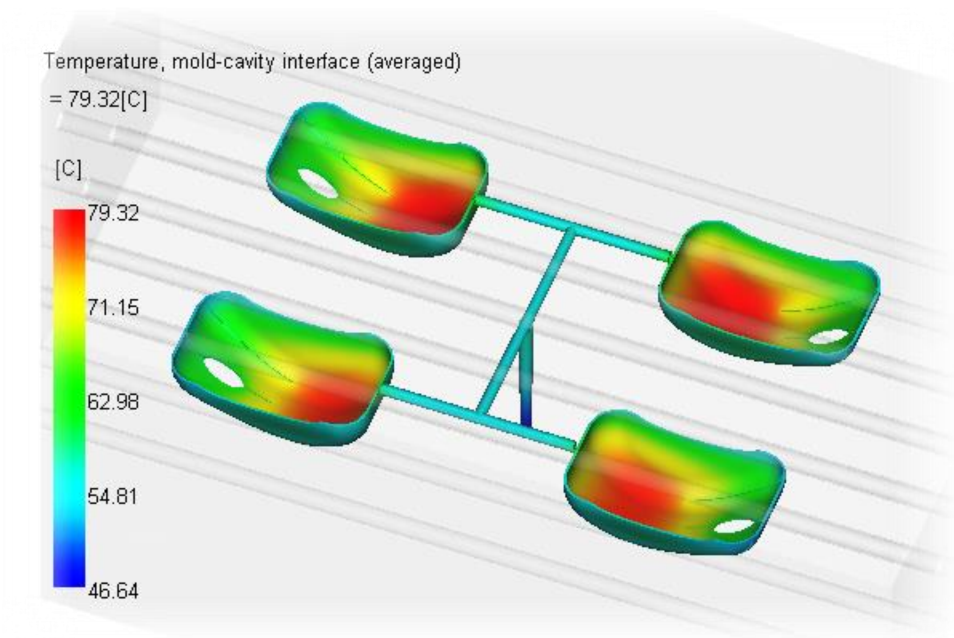


Finite Element Analysis (FEA) of Cooling

The second method is to use a cooling simulation program to estimate the cooling time necessary for the part to reach ejection temperature.

The image at the right shows the results of a cooling time analysis using Moldflow software.





Confirm the Optimized Cooling

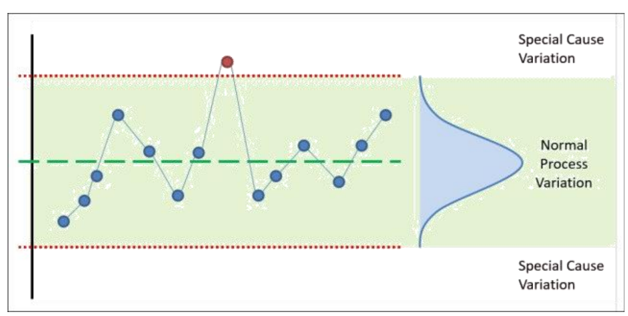
The Part Temperature at Ejection is a critical measurement of variation in both melting and cooling. If the part temperature at Ejection is consistent it is a good sign that the balance of melt temperature and mold cooling are consistent.

Digital Thermometer with Surface Probe

Infrared Thermometer

Infrared Heat Camera

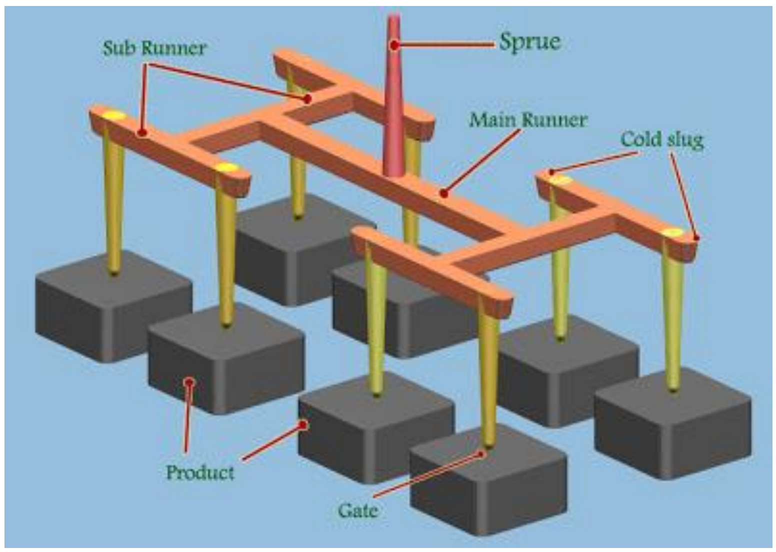
Check: Observe and record the Part Ejection Temperature periodically for the duration of the run.

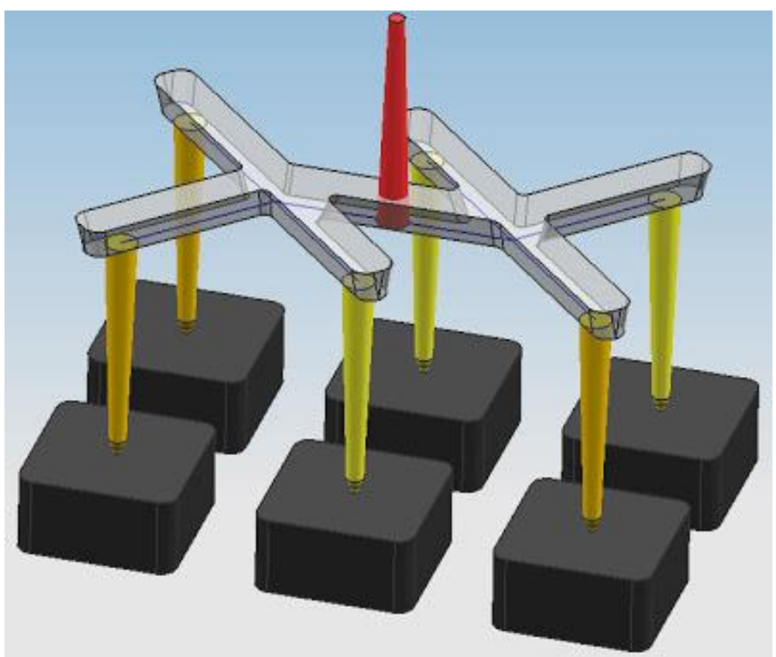


Balance

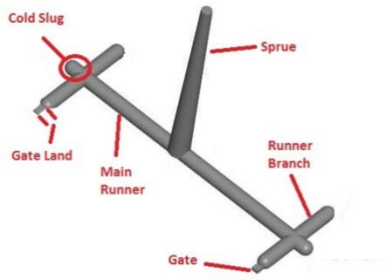
For all parts of a multi-cavity mold to have the same dimensions it is necessary that they all:

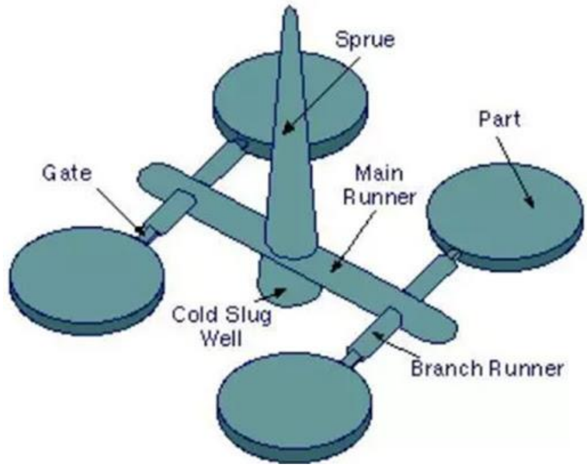
1. **fill at the same time**
2. **fill at the same pressure**
3. **and all cool the same**

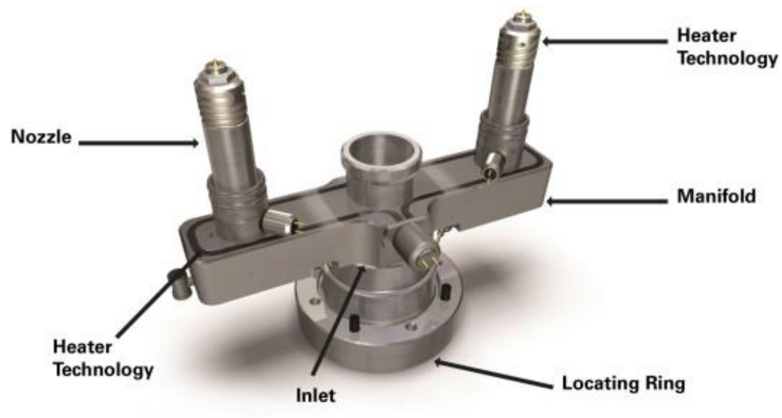




Balance of Fill, Runner Balance







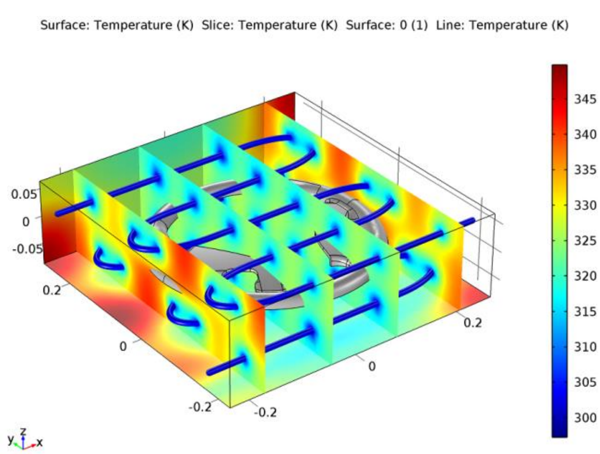
Watering a Mold

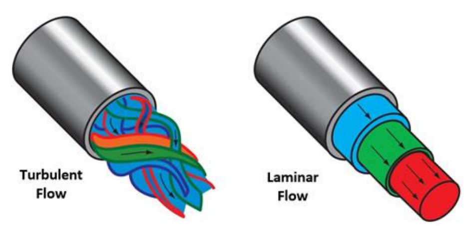
Hardware

There are a limitless number of options for how to connect water to a mold. Some of the most common are quick connects, manifolds, flow controllers.

Important Cooling Issues

It is extremely important to provide cooling to a mold as equally as possible. It is impossible to totally equally cool the mold and the parts produced. Since cooling the mold and parts equally is impossible, all efforts should be made to cool the mold consistently. Inlets and outlets should be clearly marked. When possible, inlets and outlets should be plumbed so the mold cannot be watered incorrectly. Color coding hoses is a common practice. This process is referred to as "Poka-yoke". The temperature and flow rate to each mold cooling circuit should be controlled, whenever possible.





Cooling, Warp and some important things to remember

Controlling warp

Equal cooling must be provided to both sides of the part.

- Cooling water temperature
- Cooling water flow rate
- Amount of cooling per surface area
- Cooling problems due to ejector pins, the sprue, etc.

"Equal cooling rarely produces equal cooling"

Adjusting for warp

Check the Balance of Fill

Check the Balance of Fill in Multicavity Molds

Multi-cavity molds may be balanced or unbalanced. The balance of fill experiment shows the degree of imbalance in a multi-cavity mold if any.

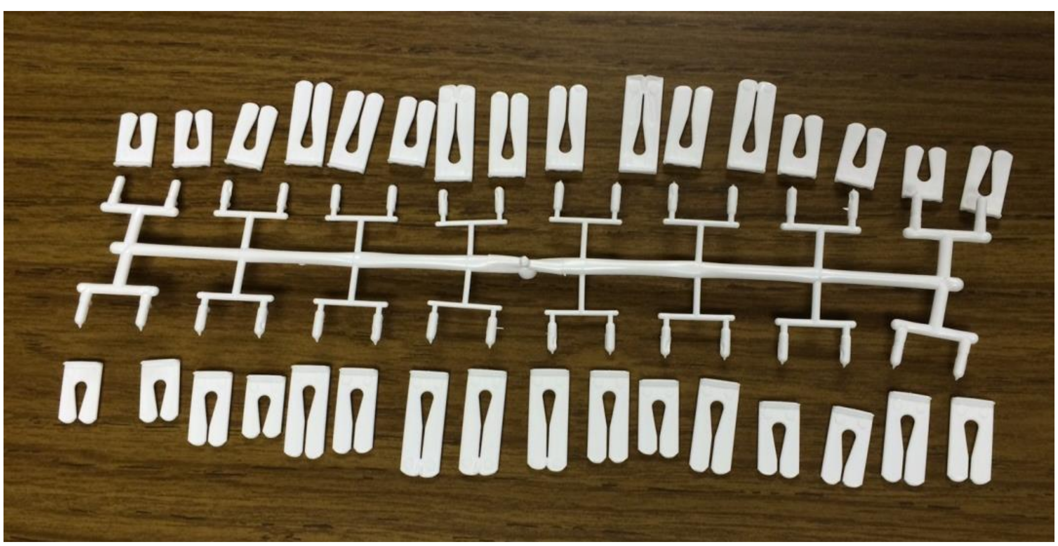
Balance of Fill Analysis

<u>Background</u>

The balance of fill analysis was created to evaluate the thermal and flow balance of the plastic distribution system of the mold, commonly called the runner system. This includes both hot and cold runner designs from the sprue to the area where the part is formed. This test is only intended for use on tooling that molds three or more like parts and has a symmetrically balanced runner system.

Runner balance can be quantified with two measures. <u>Degree of imbalance</u> provides a measure for each cavity of the variation from the heaviest cavity. <u>Overall imbalance</u> provides a measure for each cavity of the variation from the average of all cavities.

The cavity-to-cavity weight difference is an indicator of the quality of workmanship, precision of design, and accuracy of the manufacture of the melt distribution system. It is critical to have the flows balanced to each cavity or the part-to-part variation may be large, therefore making process capability unachievable.



Part #	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Part Wt.																

Part #	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	32
Part Wt.																

Procedure:

1. Obtain part for the balance of fill analysis in one the following ways.
   a. Get parts from the instructor that were previously molded.
   b. Mold consecutive "fill part" (95-98%) shots from a steady-state Injection Molding process (no assignable causes present).
   NOTE: 10 consecutive shots is an ideal sample size although this analysis can be used for any number of shots.

2. Measure the weight of each part using the most accurate scale available. Average the sample weights for each cavity.

3. For each cavity, use the following formula to calculate the degree of imbalance:

Degree of Imbalance (%) = $$\frac{(W_f - W_n)*100}{W_f}$$

where W_f = weight of heaviest cavity
W_n = weight of cavity n
n = each cavity number

4. Use the following formula to calculate overall imbalance:

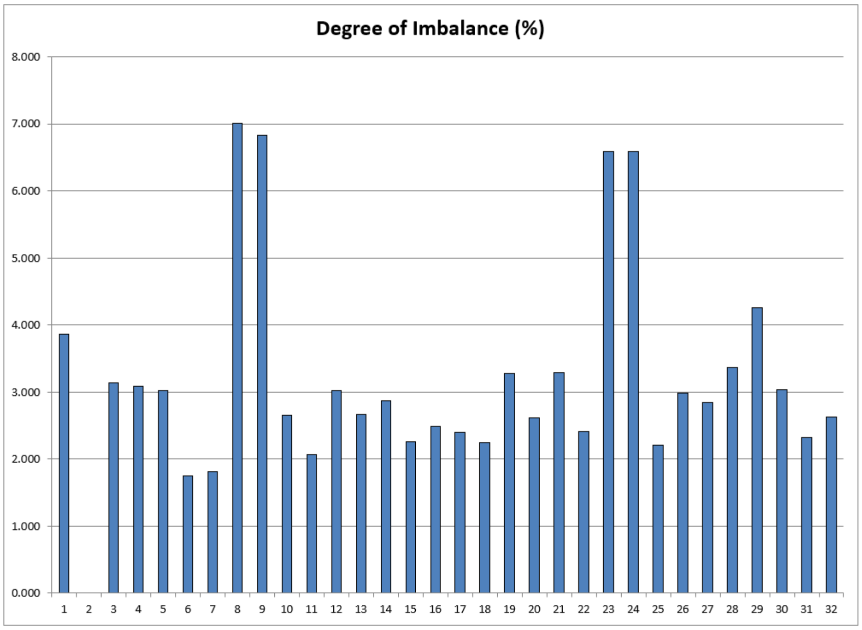
Overall Imbalance (%) = $$\frac{(W_a - W_n)*100}{W_a}$$

where W_a = average weight of all cavities
W_n = weight of cavity n
n = each cavity number

G.S. 10 Sec Hold 21,858			
Cavity Number	Part Weight (g)	Degree of Imbalance (%)	Overall Imbalance (%)
1	0.6927	3.858	0.7055
2	0.7205	0.000	-3.2794
3	0.6979	3.137	-0.0399
4	0.6983	3.081	-0.0972
5	0.6987	3.026	-0.1545
6	0.7079	1.749	-1.4733
7	0.7074	1.818	-1.4016
8	0.6700	7.009	3.9594
9	0.6713	6.829	3.7731
10	0.7014	2.651	-0.5416
11	0.7056	2.068	-1.1436
12	0.6987	3.026	-0.1545
13	0.7013	2.665	-0.5272
14	0.6998	2.873	-0.3122
15	0.7042	2.262	-0.9429
16	0.7026	2.484	-0.7136
17	0.7032	2.401	-0.7996
18	0.7043	2.248	-0.9573
19	0.6969	3.276	0.1035
20	0.7017	2.609	-0.5846
21	0.6968	3.289	0.1178
22	0.7031	2.415	-0.7853
23	0.6730	6.593	3.5294
24	0.6730	6.593	3.5294
25	0.7046	2.207	-1.0003
26	0.6990	2.984	-0.1975
27	0.7000	2.845	-0.3409
28	0.6962	3.373	0.2038
29	0.6898	4.261	1.1212
30	0.6986	3.040	-0.1402
31	0.7038	2.318	-0.8856
32	0.7016	2.623	-0.5702

Degree of Imbalance

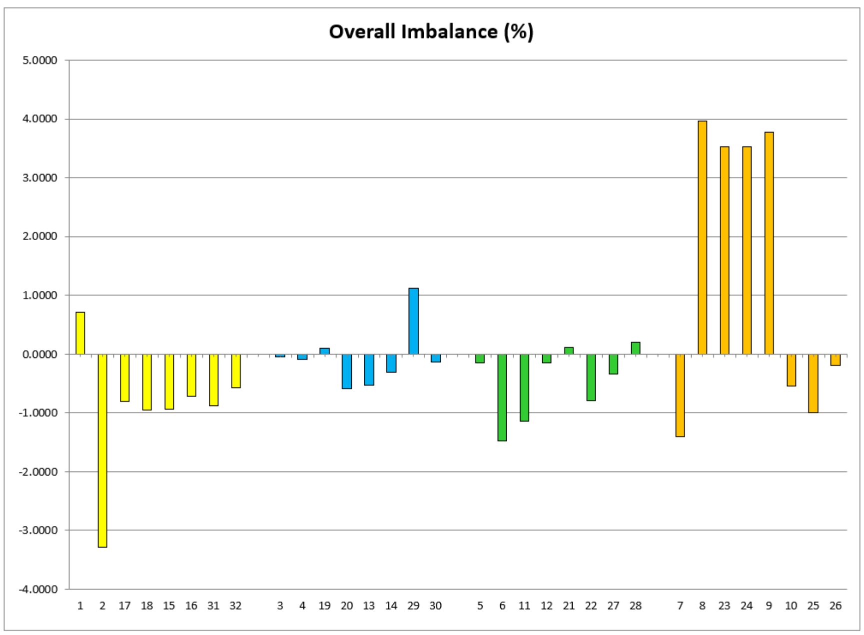
5. Produce a graph of degree of imbalance (%) like the following:



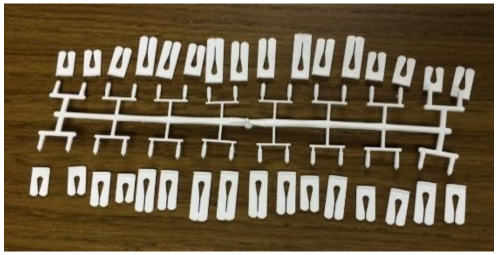
Overall Imbalance

6. Using the part runner, group cavities into "like groups" based on flow balance.

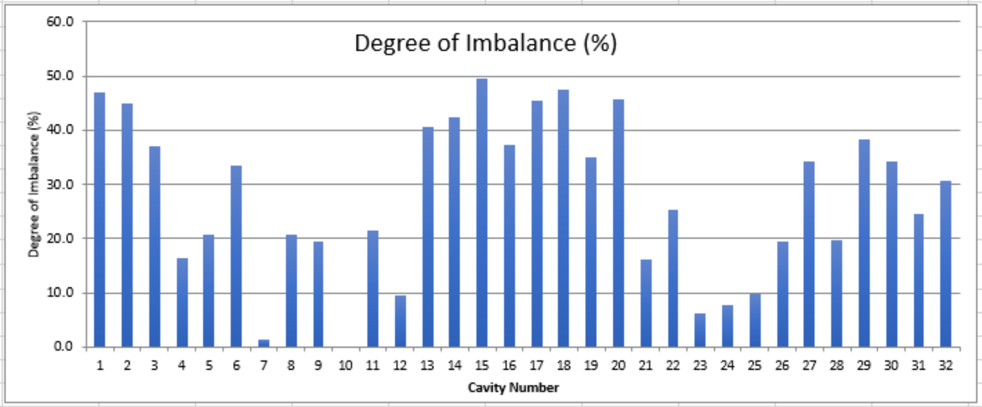
7. Build another table and graph using the overall imbalance data. The graph should look like the one below. Please group cavities in "like groups" for this analysis.

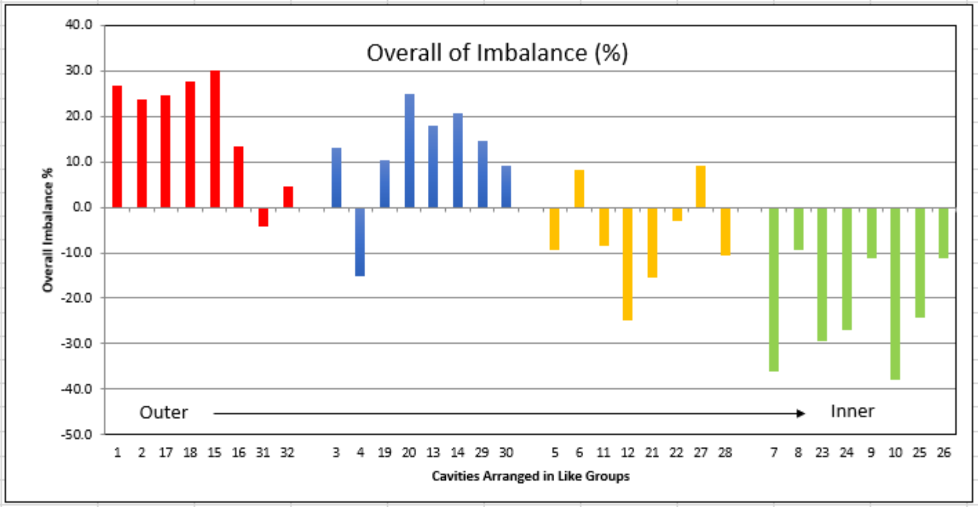


Balance of Fill Example 2



G.S. 10 Sec Hold 21,858					
Cavity Number	Part Weight (g)	Degree of Imbalance (%)	Overall Imbalance (%)	Cavity Number	Overall Imbalance (%)
1	0.3372	46.9	26.7	1	26.7
2	0.3500	44.8	23.9	2	23.9
3	0.3997	37.0	13.1	17	24.6
4	0.5297	16.5	-15.2	18	27.5
5	0.5035	20.7	-9.5	15	30.2
6	0.4226	33.4	8.1	16	13.3
7	0.6262	1.3	-36.2	31	-4.2
8	0.5036	20.6	-9.5	32	4.4
9	0.5116	19.4	-11.3		
10	0.6346	0.0	-38.0	3	13.1
11	0.4981	21.5	-8.3	4	-15.2
12	0.5737	9.6	-24.8	19	10.2
13	0.3776	40.5	17.9	20	24.9
14	0.3650	42.5	20.6	13	17.9
15	0.3209	49.4	30.2	14	20.6
16	0.3984	37.2	13.3	29	14.7
17	0.3465	45.4	24.6	30	9.1
18	0.3332	47.5	27.5		
19	0.4127	35.0	10.2	5	-9.5
20	0.3451	45.6	24.9	6	8.1
21	0.5314	16.3	-15.6	11	-8.3
22	0.4733	25.4	-2.9	12	-24.8
23	0.5946	6.3	-29.3	21	-15.6
24	0.5846	7.9	-27.2	22	-2.9
25	0.5717	9.9	-24.4	27	9.2
26	0.5116	19.4	-11.3	28	-10.7
27	0.4176	34.2	9.2		
28	0.5089	19.8	-10.7	7	-36.2
29	0.3922	38.2	14.7	8	-9.5
30	0.4179	34.1	9.1	23	-29.3
31	0.4789	24.5	-4.2	24	-27.2
32	0.4393	30.8	4.4	9	-11.3
				10	-38.0
Lead Cavity	0.6346			25	-24.4
Average	0.4597			26	-11.3





Decoupled 2 (D2) vs, Decoupled 3 (D3) Molding

The basic philosophy behind decoupled molding is the separation, or decoupling, of the filling portion of an injection molding cycle from the balance of the injection molding cycle. There are two decoupled molding styles being used in industry today. They are referred to as "D2" and "D3" molding set-up, validation, and troubleshooting techniques.

D2 molding, short for decoupled 2 molding, separates the filling portion of the cycle from the pack and hold portion of the cycle. D3 molding, short for decoupled 3 molding, separates the filling portion of the cycle from the pack portion of the cycle which is separated from the hold portion of the cycle. The "tool" used to separate the portions of the cycle is called transfer.

Decoupled Set-Up, Validation, Troubleshooting

Decoupled 2 Molding ("D2 Molding") = Fill – transfer – Pack and Hold

Fill	»	Pack	Hold

Decoupled 3 Molding ("D3 Molding") = Fill – transfer – Pack - transfer – Hold

Fill	»	Pack	»	Hold

Velocity based Fill and Pressure based Pack and Hold

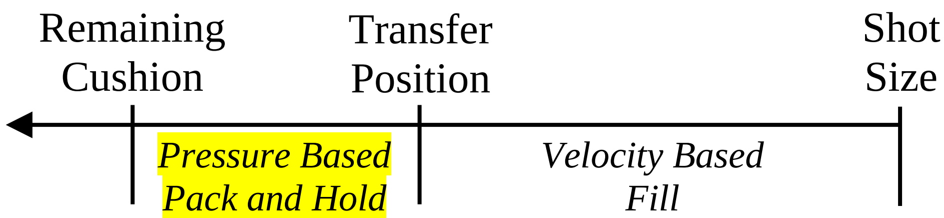
Transfer (often called V to P transfer) is necessary because the settings for fill are based on the velocity of the screw during injection. The pressure needed to drive the screw at the requested speed is tied to the viscosity (thickness) of the material and is relatively less important.

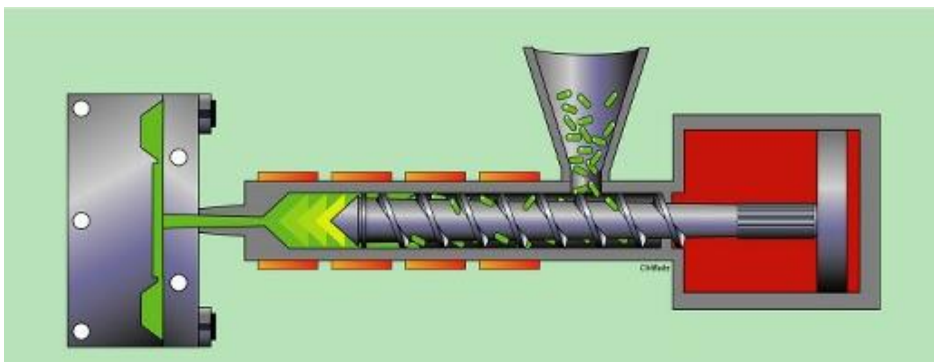
The machine settings for pack and hold are based on the pressure ultimately developed in the cavity. During pack and hold the screw should barely move. The velocity of this movement is less important than the ultimate pack pressure.

D2 fill to pack and hold transfer is typically found on the Injection Molding machine. D3 fill to pack transfer is the same machine-based transfer. D3 pack to hold transfer is based on data from a cavity pressure sensor placed in the mold. Decoupled 3 molding is just a more sophisticated method of controlling the final cavity pressure if a cavity pressure sensor is available in the mold. Types of transfer will be discussed later.

Controlling Dimensions with Cavity Pressure

The key in Decoupled Molding is separating the filling phase, with the associated fill pressure, from the packing phase and the associated packing pressure.





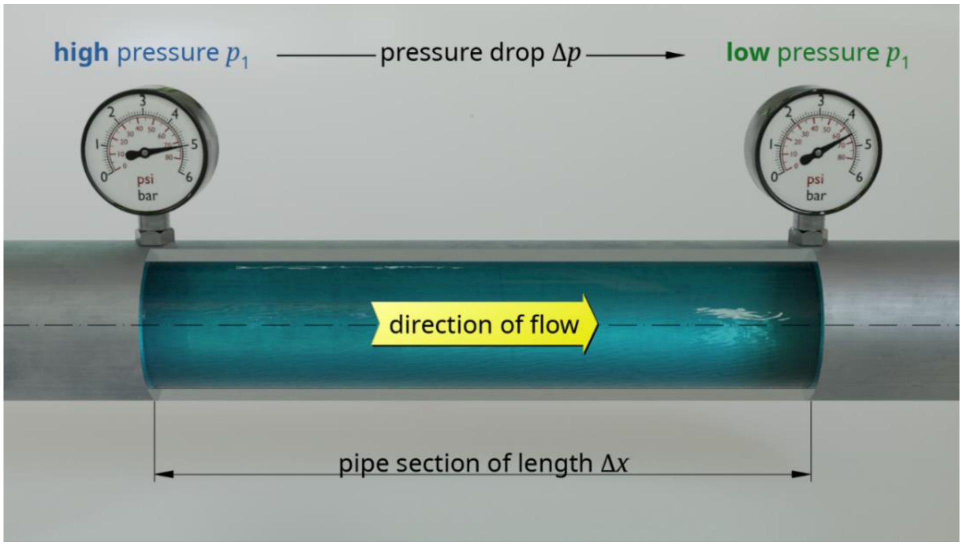
Decoupled 2: No Cavity Pressure Sensors

$\text{Part Dimensions} \leftarrow \text{Pressure}_{\text{Cavity}} \leftarrow \text{Pressure}_{\text{Pack}}$

Decoupled 3: With Cavity Pressure Sensors

$\text{Part Dimensions} \leftarrow \text{Pressure}_{\text{Cavity}}$

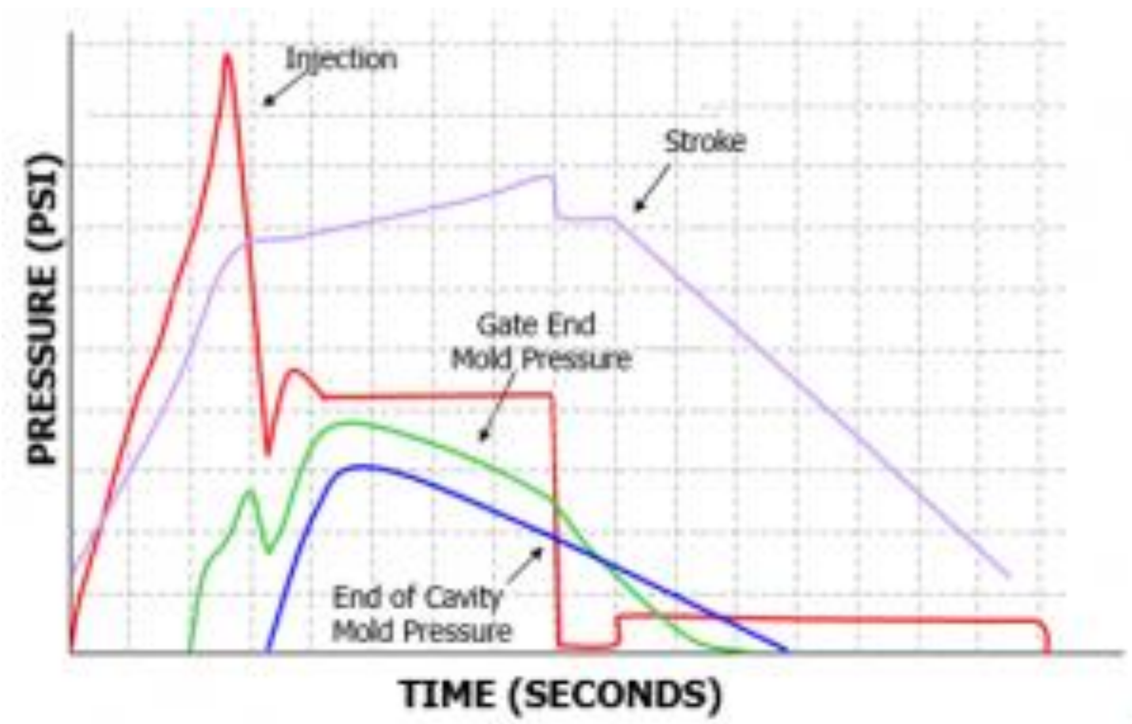
"More pressure equals bigger parts"



ΔP = Pressure loss

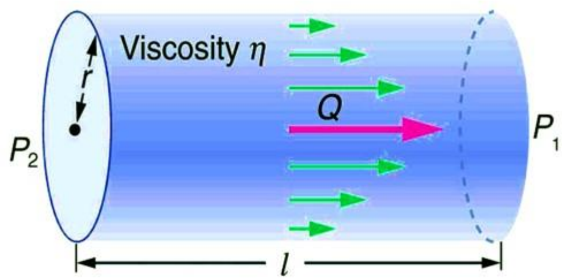
↓

Part Dimensions ⬅ $\text{Pressure}_{\text{Cavity}}$ ⬅ $\text{Pressure}_{\text{Pack}}$



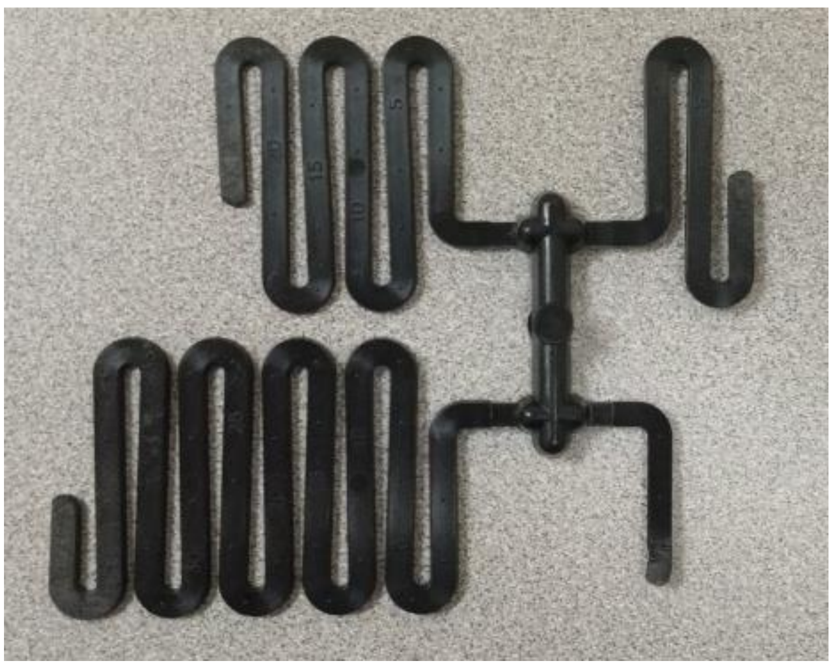
Hagen-Poiseuille Law

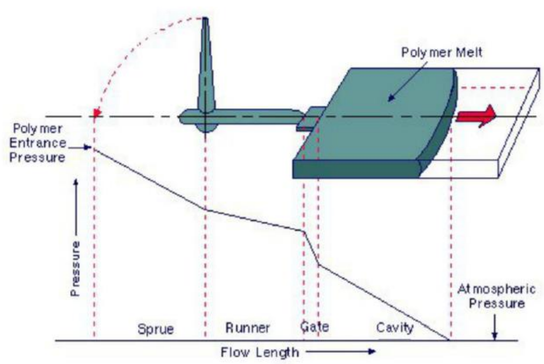
ΔP = Pressure loss

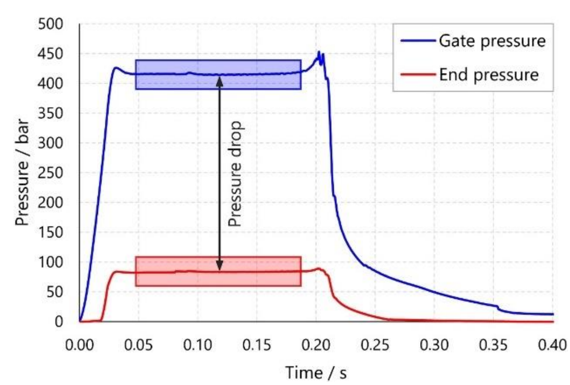


$$\Delta P = \frac{8\mu QL}{\pi r^4}$$

where: ΔP = Pressure loss (lb./in^2) (aka: PSI)
μ = viscosity of the polymer melt (lb. sec/in^2)
Q = volumetric flow rate (in.^{3}/sec)
L = the length the melt is flowing (in.)
r = the radius of the runner segment (in.)
8 and π are numerical constants (no dimensions)







Quick Checks on Running Cycles

Not sure if the current cycle is set up correctly? Here are some cycle quick checks to assure the current running cycle is decoupled correctly.

<u>Quick Check that will not affect Parts Produced.</u>

✓ Fill Pressure Checks

1. Increase Fill Pressure and observe Fill Time. If the cycle is not pressure limited nothing will happen.
2. Observe a graph of Set Fill Pressure and Actual Fill Pressure vs. Screw Position. If the two pressure lines touch. The process is pressure limited.

✓ Material Viscosity Check

- Multiply the Pressure at Transfer times the Fill Time. This number should not change throughout a process run.

<u>Quick Checks that will affect Part Produced.</u>

✓ Dose Check (screw travel from Shot Size to Transfer).

- Set all pack pressures and hold times to 0 for one cycle. A shot with all parts short should be produced. The fullest part (the lead part) should be 95-98% full. Recouple by setting pack pressures and hold times back to the original settings.

Important Variable Check List

	Value	Variable	Data from the …
		Fill Pressure Limit	Pressure Limited Experiment(s)
		Injection Speed	Velocity Optimization Experiment
		Shot Size	Short Shot Experiment
		Transfer Position	Short Shot Experiment
		Cushion	Short Shot Experiment
		Screw Speed	Tacking Temperature Experiment
		Backpressure	Tacking Temperature Experiment
		Barrel Heats	Tacking Temperature Experiment
		Hold Time	Gate Seal Experiment
		TCU Temperature	Ejection Temperature / Warp
		Coolant Flow Rate	Ejection Temperature / Warp
		Cooling Time	Ejection Temperature / Warp
		Melt Temperature	30/30/30 Air Shot Melt Temp
		Pack Pressure	Maximizing C_{PK}
		Multi-Cavity Consistency	Balance of Fill Experiment

The Parts of an Injection Molding Cycle - The Answers

Before we can begin the optimization and troubleshooting process we must agree on a method of "scientifically" setting up a molding machine. We will begin by discussing the parts of an injection molding cycle followed by a detailed discussion of a set-up technique known as "Decoupled Molding".

Let's begin by looking at the parts of an Injection Molding Cycle. Run the animation im.exe. Name as many parts of the cycle as you can as the animation runs.

- **Start at:** 1. The mold closes. (the industry standard beginning)

2. The Injection Unit moves forward. (sprue break) *(if selected)*

3. Injection. (Fill)

4. Injection. (Transfer)

5. Injection. (Pack and Hold)

6. Make the next shot. (screw recovery, aka: Plastication)

7. Suck back. (aka: melt decompression) *(if selected)*

8. The Injection Unit moves away. (sprue break) *(if selected)*

9. The Mold Opens.

10. Core pull out. *(if selected)*

11. Eject part.

12. Core pull in. *(if selected)*

The Power of Decoupled Molding

1. Decoupled molding provides a data-driven method for converting all the variables associated with injection molding to constants.
2. Everyone everywhere develops the same cycle regardless of education, experience, or location.

Key Things to Remember

1. More pressure equals bigger parts.
2. Heating to and Cooling from Melt Temperature will change the dimensions of a part.
3. Cooling is typically the longest part of an injection molding cycle. If you are looking to reduce cycle time, start here.
4. It is important to control the rate of cooling both in the mold and after ejection and the part Ejection Temperature.
5. The gate almost always cools before the part.
6. It is better to fill on pack than pack on fill.
7. The 95% part (the Lead Part) should be the biggest part in a decoupled multi-cavity shot.
8. Faster injection produces more orientation.
9. More orientation reduces viscosity.
10. Never run zero cushion.

Keep an Eye on the Money, the Answers and Solution

Yes, for some, Math is a “four-letter” word. The solutions below show how by keeping track of the dimensions, like hours per day, we can make the math much easier. The first table shows only the dimensions for the problems in activity 1. There is the price of the parts (net over material cost) in dollars per part. The number of mold cavities, which is the number of parts produced per cycle. Next is the cycle time, expressed in cycles per second. Finally, there are the factors for converting seconds in a year. From the math perspective, it is important to see that all of the units “cancel out” except for dollars and year. Our answer is then in dollars per year. The first table is the “units only” table.

Annual Value Of Parts	=	Price (net over material cost)	×	Mold Cavities	÷	Cycle time	×	sec/ min	×	min/ hr.	×	hrs./ day	×	days/ week	×	weeks/ year
$ / **year**	=	**$** / ~~part~~	~~×~~	~~parts~~ / ~~cycle~~	~~÷~~	~~cycle~~ / ~~sec~~	~~×~~	~~sec~~ / ~~min~~	~~×~~	~~min~~ / ~~hour~~	~~×~~	~~hours~~ / ~~day~~	~~×~~	~~days~~ / ~~week~~	~~×~~	~~weeks~~ / **year**

The next table is the “units only” table with the “given” values plugged in for a 20-second cycle.

Annual Value Of Parts	=	Price (net over material cost)	×	Mold Cavities	÷	Cycle time	×	sec/ min	×	min/ hr.	×	hrs./ day	×	days/ week	×	weeks/ year
???		$0.06		64		**20**		60		60		24		7		50
$ / **year**	=	$ / ~~part~~	~~×~~	~~parts~~ / ~~cycle~~	~~÷~~	~~cycle~~ / ~~sec~~	~~×~~	~~sec~~ / ~~min~~	~~×~~	~~min~~ / ~~hour~~	~~×~~	~~hours~~ / ~~day~~	~~×~~	~~days~~ / ~~week~~	~~×~~	~~weeks~~ / **year**

The next table is the “solved” table with the “given” values plugged in for a 20-second cycle.

Annual Value Of Parts	=	Price (net over material cost)	×	Mold Cavities	÷	Cycle time	×	sec/ min	×	min/ hr.	×	hrs./ day	×	days/ week	×	weeks/ year
$5,806,080		$0.06		64		**20**		60		60		24		7		50
$ / **year**	=	**$** / ~~part~~	~~×~~	~~parts~~ / ~~cycle~~	~~÷~~	~~cycle~~ / ~~sec~~	~~×~~	~~sec~~ / ~~min~~	~~×~~	~~min~~ / ~~hour~~	~~×~~	~~hours~~ / ~~day~~	~~×~~	~~days~~ / ~~week~~	~~×~~	~~weeks~~ / **year**

The next table is the “solved” table with the “given” values plugged in for a 17-second cycle.

Annual Value Of Parts	=	Price (net over material cost)	×	Mold Cavities	÷	Cycle time	×	sec/ min	×	min/ hr.	×	hrs./ day	×	days/ week	×	weeks/ year
$6,830,682		$0.06		64		**17**		60		60		24		7		50
$ / **year**	=	**$** / ~~part~~	~~×~~	~~parts~~ / ~~cycle~~	~~÷~~	~~cycle~~ / ~~sec~~	~~×~~	~~sec~~ / ~~min~~	~~×~~	~~min~~ / ~~hour~~	~~×~~	~~hours~~ / ~~day~~	~~×~~	~~days~~ / ~~week~~	~~×~~	~~weeks~~ / **year**

THINGS TO REMEMBER: