Working with the Machine

LESSON 3: Optimizing the Machine Controls

Lesson Three: Optimizing Machine Controls

Troubleshooting injection molding problems can be very compli-cated and confusing. There are many interrelated variables. When one variable is changed, it can affect many others. There are some techniques, however, that tend to move toward a solution faster than others. This lesson discusses how to start the process, and points out some of the best steps to take when facing molding problems.

Objectives of Lesson Three

1. Define the goals and objectives of part production
2. Learn how to make the first full shot
3. Learn how to minimize cycle time
4. Learn how to find the best pressures and speeds
5. Learn how to adjust barrel and mold temperatures

Objective One

Part Production Objectives

The Goal of the Process

The goal of the process is to create quality parts that are dimen-sionally accurate, using the shortest possible cycle time. Ideally, the parts should be produced with temperature, pressure, and speed settings in the middle of their acceptable processing win-dows.

Quality Parts

Obviously, the overall goal is to make quality parts economically. Quality parts have the proper surf ace finish, acceptable weld lines, and dimensional accuracy. Quality levels are set by the quality control department, and identified in specification sheets.

Dimensional Accuracy

To the molding technician, dimensional accuracy is largely a matter of controlling part shrinkage and packing in the cavity. Each plastic has its own shrinkage rate. The technician should have a general idea of how much shrinkage is expected. This will help define how much change can be made with the processing controls. The mold cavities have already been built oversize to accommodate the expected part shrinkage.

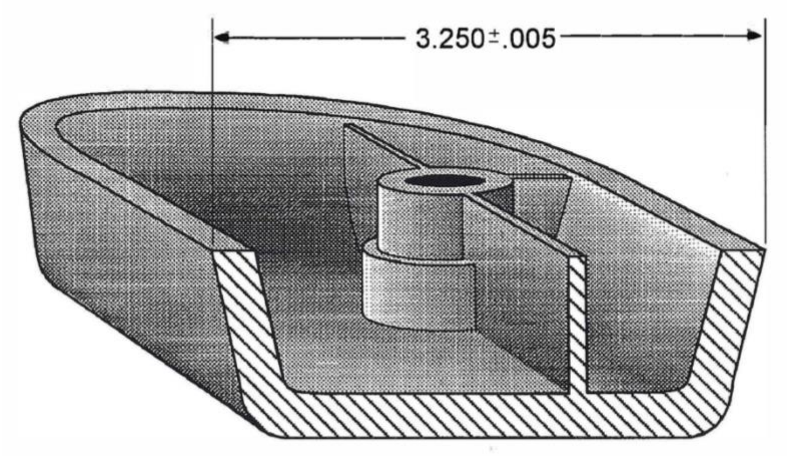

Figure 1 - Dimensional Tolerance

Many plastic parts, like that in Figure 1, have tight dimensional tolerances, making accurate process control very important.

For the molding technician, there are a number of related process parameters that will help control part dimensions. The most important parameters are injection pressures, and boost and hold time.

Mold temperature and cooling time also affect part size. Increas-ing injection pressure packs more plastic in the cavity. Long boost and holding times continue to pack extra plastic into the cavity as the material shrinks away from the walls.

The mold cavity acts like a shrink fixture; the longer the parts are in the mold, the less total shrinkage will occur. If the parts are ejected hot, they will shrink sitting in the bin or on the table. Excessive cooling times are a problem, however, because they make the cycle too long. A compromise often must be made.

Mold temperature works in a similar way to cooling time. The colder the mold, the quicker the part solidifies, and the less chance it has to cool in its unrestrained condition outside the mold. Also, colder molds generally reduce the overall time needed to cool the parts.

Setup Time and Cycle Time

The molding technician needs to develop thought-out, efficient setup and troubleshooting skills when adjusting the machine controls. Much time can be lost, and many wasted parts can be produced, if the process is all trial and error without applying much of the available molding theory.

Figure 2 illustrates how a cycle can be broken up into its components, which can then be analyzed to achieve the shortest cycle time. The technician should always be concerned about adjusting and monitoring the cycle time components. One extra second per cycle can add up to many hours of wasted machine time per month.

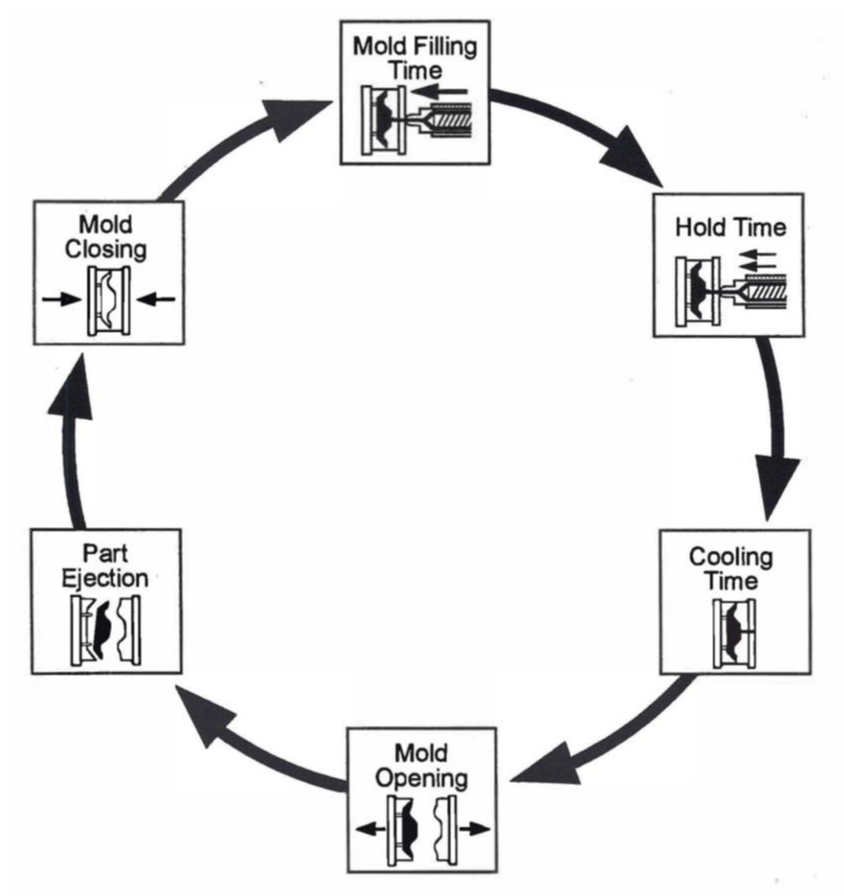

Figure 2 - The Molding Cycle

Process Windows

One of the reasons that the molding process is so complicated is because each of the process variables (temperature, pressure, speed, etc.) has a process window. A process window is a range that the process parameter can fluctuate in without affecting the quality of the part.

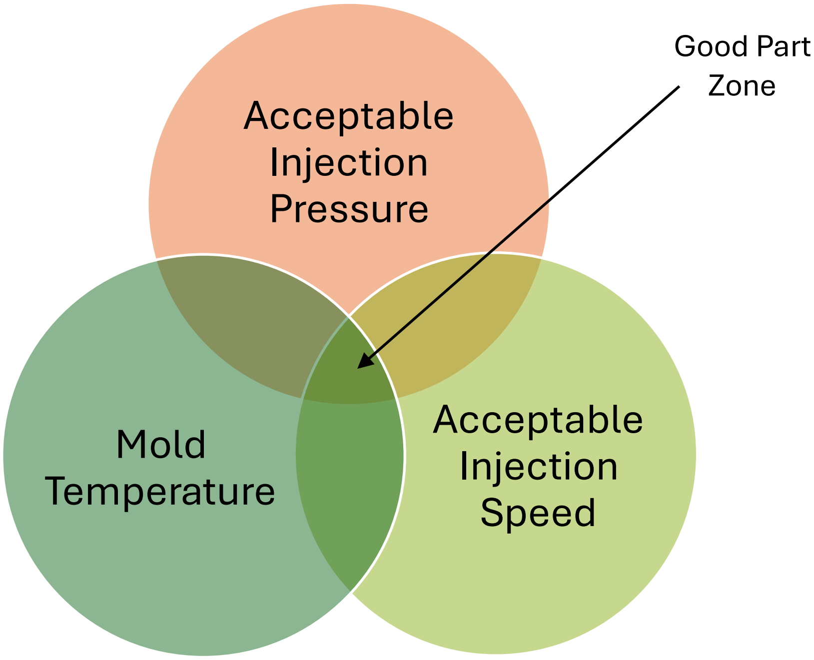

Figure 3 – Process Window

Figure 3 shows how molding processes have windows that overlap. Where all three windows overlap, good parts are pro-duced. Each of the process parameters is related to the others. When one parameter changes, it moves the process window for all the related parameters.

Example:

Machine A is making good polypropylene parts with the follow-ing conditions:

Barrel Temperature .... 450°F

Mold Temperature ....120°F

Injection Transfer Pressure ....1250 psi

Water Outlet Lines ....10800 psi

Injection Velocity 2.0in/sec

Hold Time ....12 sec

Cool Time .... 21 sec

If more regrind (ground up runners) were added to the melt, it may become less viscous. This means the melt would be runnier, causing it to flow easier. Regrind often lowers the viscosity, because the regrind molecules are typically shorter than the virgin material.

The new runnier melt will flow faster into the mold. It may cause the parts to flash. (Seeing the flash, the technician may reduce the boost pressure.) Lowering the boost pressure could cause an increase in the fill time that could result in warpage. To keep the fill time constant, the technician may decide to raise the injection speed to re-stabilize the process.)

Notice that, even though only one element was changed, (adding more regrind to the melt) the overall process may have to be changed considerably to get the machine making good parts again. Once one component of the process changes, it may require many other changes before the process is back in the window again.

Establish Processing Windows

Each parameter operates in a window of acceptable values. Each window is dependent on the other parameters. The overall process will be safer, surer, and more repeatable if all parameters are operating somewhere in the middle of their windows. This way, a change in one parameter is not as likely to throw the entire process out of tolerance. Staying in the middle of the windows also makes the process more repeatable. There is more of a margin for error.

Exercise One

Process Variables

Find a job in your shop that seems to be running consistently good parts. Record all the settings for speed, pressure, temperature, and timing you can find.

Speed	Pressure	Temperature	Timing

Instructor Date

Techniques for Adjusting the Process

Evolutionary versus Revolutionary

There are two major techniques for adjusting the injection mold-ing machine. These techniques can be used during setup, or while fine tuning the process. The two techniques are called evolutionary (long and slow) or revolutionary (short and quick). Each has its advantages and disadvantages. Either approach can be applied successfully in the appropriate situation.

Evolutionary

The evolutionary method involves making very small adjust-ments to the process (Figure 4) and recording the results. In most cases, this means running the, machine for some time while the process stabilizes with the new settings. This method requires that the molding technician carefully record the process condi-tions, what has changed, and the result of the change.

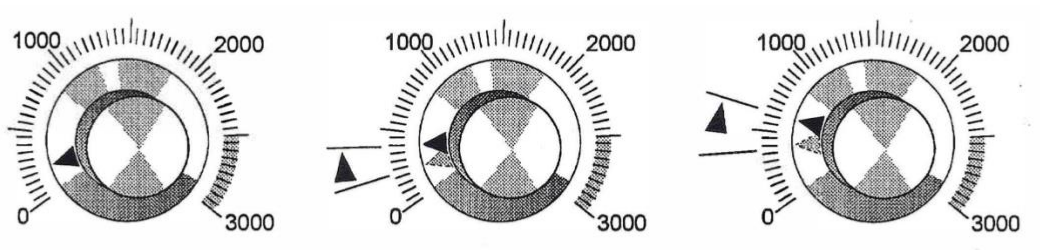

Figure 4 - Evolutionary Adjustments

This method also requires that the process be fairly well con-trolled, and that the performance of the machine can be measured. The evolutionary method is very useful in characterizing the process and the machine. It works best in long runs with the same mold. This method does not work well if the machine is currently making scrap, or the process needs a major adjustment.

Revolutionary

The revolutionary method involves making an extreme adjust-ment in an attempt to find the outer bounds of the process window (Figure 5). Extreme adjustments produce results quickly, but they are less controlled. This method requires the technician to make an extreme adjustment, then back off the change until the process is in the proper window.

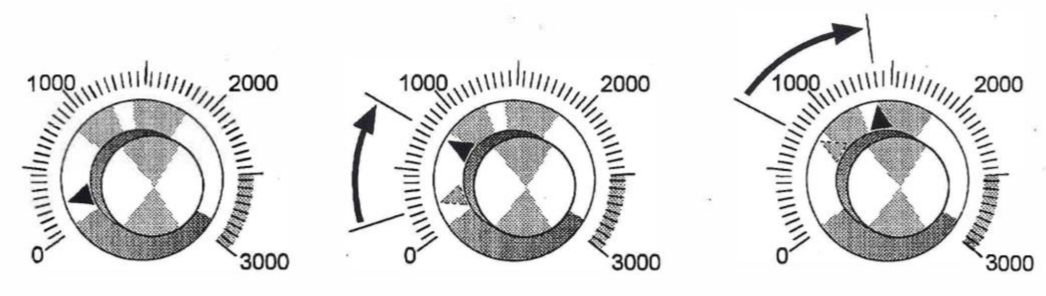

Figure 5 – Revolutionary Adjustments

The revolutionary method is useful when the machine is making scrap, or in the initial setup of a new job. Making scrap means that every part the machine makes is defective. All the parts must be scrapped. The extreme changes will establish the boundaries of the process window more quickly.

Sampling or Trial Runs

Usually, new molds, or used molds from new customers, have no setup sheets with them. These situations require starting from scratch in setting up the molding parameters that will lead to a good running process. Sometimes, the engineering department has prepared some guidelines to use, such as recommended barrel temperatures or estimated cycle times.

The other common situation is when an existing mold is being run on a different size machine, or one from a different manufacturer. In this case, only some of the information off the existing setup sheet, such as barrel heats, can be used without adjustment. Cycle time setting, pressure, and speed information, can sometimes be modified for use on the new machine. Either way, always gather as much information as possible to assist in the trial run process.

Technicians need to make sure that they are familiar with the internal configuration and operation of the mold. Many new or complicated molds are given to an experienced technician or a process engineer for sampling. Slides, core pulls, and delicate cores all contribute to the complexity of the mold, and require great care to be set up properly. Consult with the tooling or engineering department if necessary for information on any mold in which you are not familiar. Often, the tooling department can demonstrate any internal design conditions to the technician by opening the mold on the bench before it is hung in the machine.

Objective Two

Developing the First Shot

The following is only a quick run through of the most important steps. It is not intended to be exhaustive coverage.

Working off a setup sheet is normally simpler than sampling. Once a complete understanding of a trial run from scratch is mastered, then working off a setup sheet will seem easy.

This objective is an overview of the steps that should be taken to sample anew mold. This will help you understand the procedures and adjustments that may be necessary to run a molding machine.

Where To Start

Start with a machine that has undergone a complete setup. All the pre-start checklists should be done. All of the safety devices should be checked to make sure they are operating correctly. The barrel temperature should be up and stabilized, and the mold temperature should be checked.

The following guidelines were previously listed as suggested starting ranges for making a first shot.

Shot Size .... Large

Back Pressure .... Low

Screw Speed .... Medium

Injection Speed .... Medium

Hold Pressure .... Low

Fill Time .... Long

Hold Time .... Long

Begin by setting each cycle time component slightly longer than anticipated. Later, when the process has been further refined and stabilized, the cycle time can be optimized.

Short Shot Development

Figure 6 shows a short shot progression. When making the first few shots, it is normally more desirable to start making short shots and then fill them out than it is to start making parts which flash, and then bring them down to the proper size. There is usually more potential for mold damage or stuck parts with flashing verses short shots.

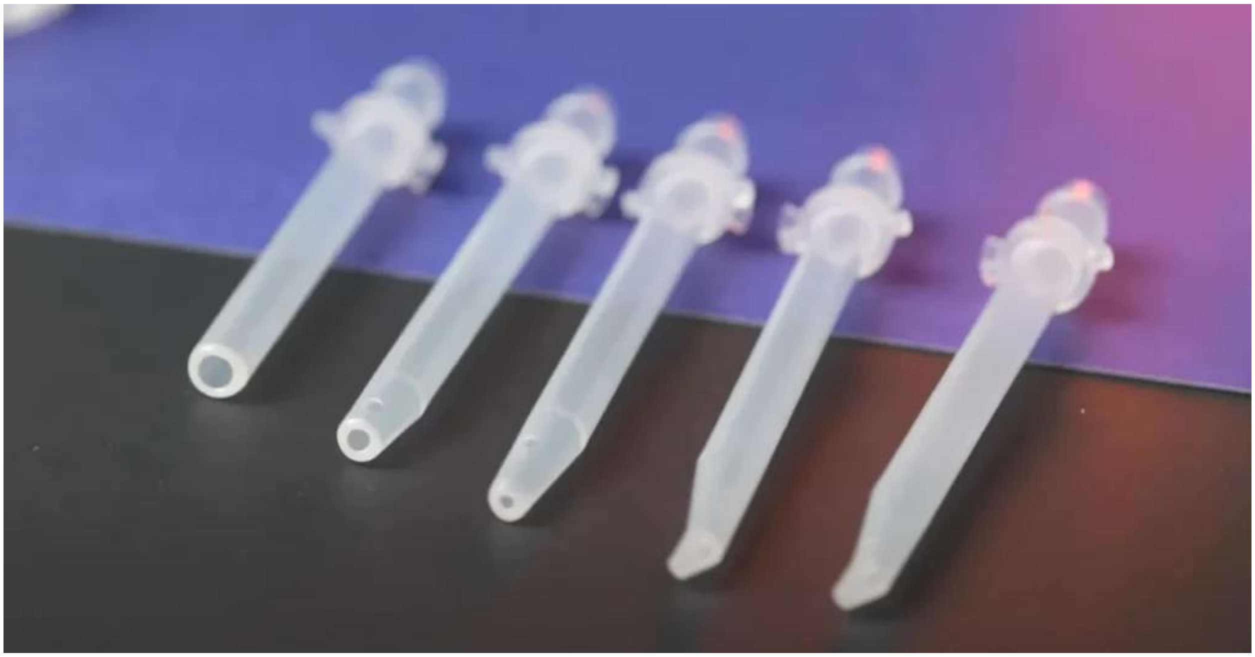

Figure 6 - Short Shot Progression

Many molding technicians like to keep things simple by only using boost pressure to develop and define the injection pressure requirements needed to fill the cavity. Others use the exact same boost and hold pressure until a full part is developed for the same reason. Either way works fine. The holding pressure may be lowered later.

The shot size should be set larger than necessary to ensure that any short shots are not due to a lack of available material. Start with a relatively low boost pressure and injection speed, and shoot the first part. Examine the degree of fill on the part, and then increase the boost pressure or injection speed. Keep increasing the injection pressure and/or speed until a full part is made. Thin areas and fins should be examined carefully to see if they have filled out properly.

Once the part has filled properly, make a note of the injection pressure and speed. Then increase the pressure slightly to ensure that complete parts will continue to be made.

With the pressure and speed now set, it is time to minimize the shot size. Reduce the shot size gradually until a 1/16 to 1/4 inch cushion is left on the shot size indicator after the cavity is filled.

Exercise Two

Cushion Size

On several machines, record the cushion size from the shot size indicator.

Machine Number	Cushion Size

Instructor Date

Objective Three

Minimizing the Cycle Time

Once a full shot has been made, the next step is to minimize the overall cycle time. Cooling time is the longest segment in the molding cycle. Cooling time directly affects part shrinkage and part dimensions. Therefore, cycle times need to be set fairly close to optimum before the other process parameters can be refined.

Minimizing First Stage Time

With the boost timer set for a longer time than necessary, take a stop watch and time how long it takes to fill the cavity. Keep your eye on the injection end of the machine, or on the shot position indicator. Start your watch when the screw begins to move towards the mold, and stop it when the screw stops moving forward. Set the first stage timer, or boost timer, about a second longer than the fill time you recorded (Figure 7).

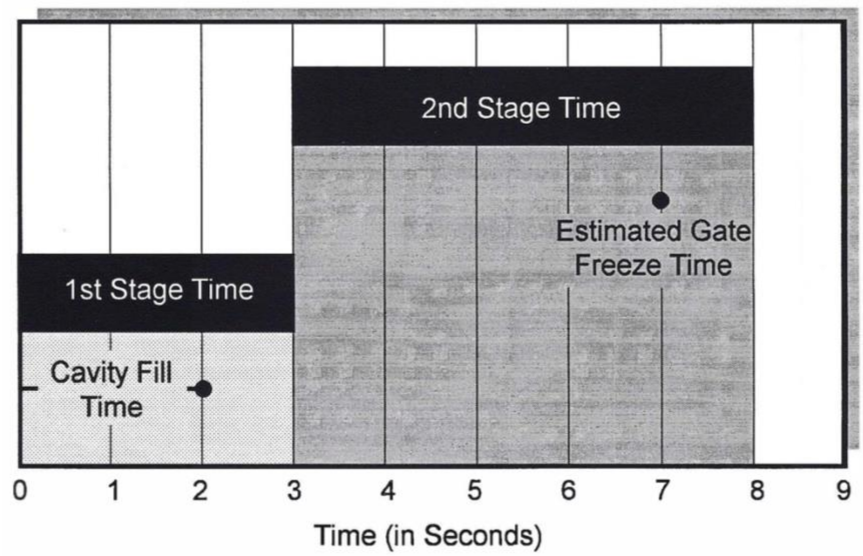

Figure 7 - Cycle Time

Minimizing Holding Time

Set the hold timer longer than necessary, and continue shooting parts while reducing the holding timer. Check each succeeding part for noticeably greater sink marks. Sink marks are depres-sions or dimples on the surface of the part. They usually show first on a wall opposite a rib or an intersecting wall. Sink marks will start occurring when the holding time has become shorter than the time it takes for the gates to freeze. This is because the plastic discharges back through the unfrozen gate and into the runner, de-pressurizing the cavity and leaving the sink marks behind. In-crease the holding time in one second increments until these major sink marks disappear, and then add an extra second or two.

Minimizing Cooling Time

The next step is to reduce the cooling time. The cooling time is too short when the parts do not solidify enough before ejection. If the part is still too hot, it will bend or tear as it separates from the core. If the cooling time is much too short, the ejector pins may punch holes in the part without ejecting it. Increase the cooling time a few seconds beyond what was obviously too short to be a reasonable setting.

It is hard to find the true absolute minimum cooling time. Parts often continue to warp or distort slightly as they cool on the bench. It is enough, at this stage of the process, to make parts that eject cleanly without obvious signs of distortion.

Minimizing Mold Open Time

On machines running in automatic mode, the mold open timer can now be reduced. The initial mold open timer setting was deliber-ately set too long. Now set the mold open timer a second longer than it actually takes to open, and eject the part cleanly. This allows the part to clear the mold before the clamp closes.

Exercise Three

Minimum Settings

On one machine, record the minimum settings for each of the following variables. Also, record what happens to the process if the variable is set shorter than the minimum.

Machine Number	What happens if you minimize the variable
Fill Time	
Hold Time	
Cool Time	

Instructor Date

Clamp Movements

Once the machine is cycling properly, the clamp stroke and ejector strokes can be reduced. Shorten the ejector stroke and clamp stroke so that the mold opens just far enough to eject the part safely. Be careful to allow enough time and room for the part to eject completely.

Screw Recovery Time

If the screw takes too long to recover, it can make the cycle time too long (Figure 8). Shorten the recovery time if the screw is still rotating when the cooling timer times out and the mold is ready to open.

There are several ways to decrease the screw recovery time. One is to increase the screw rpm. Another is to reduce the back pressure. A third is to raise the barrel heats so the screw can rotate easier. All three, though, have the potential to create other problems, so care needs to be taken when making these adjustments.

Ideally, the screw will get back just a few seconds before the mold is ready to open. It can actually be detrimental for the screw to recover too early, since this creates too much dwell time, as shown in Figure 8.

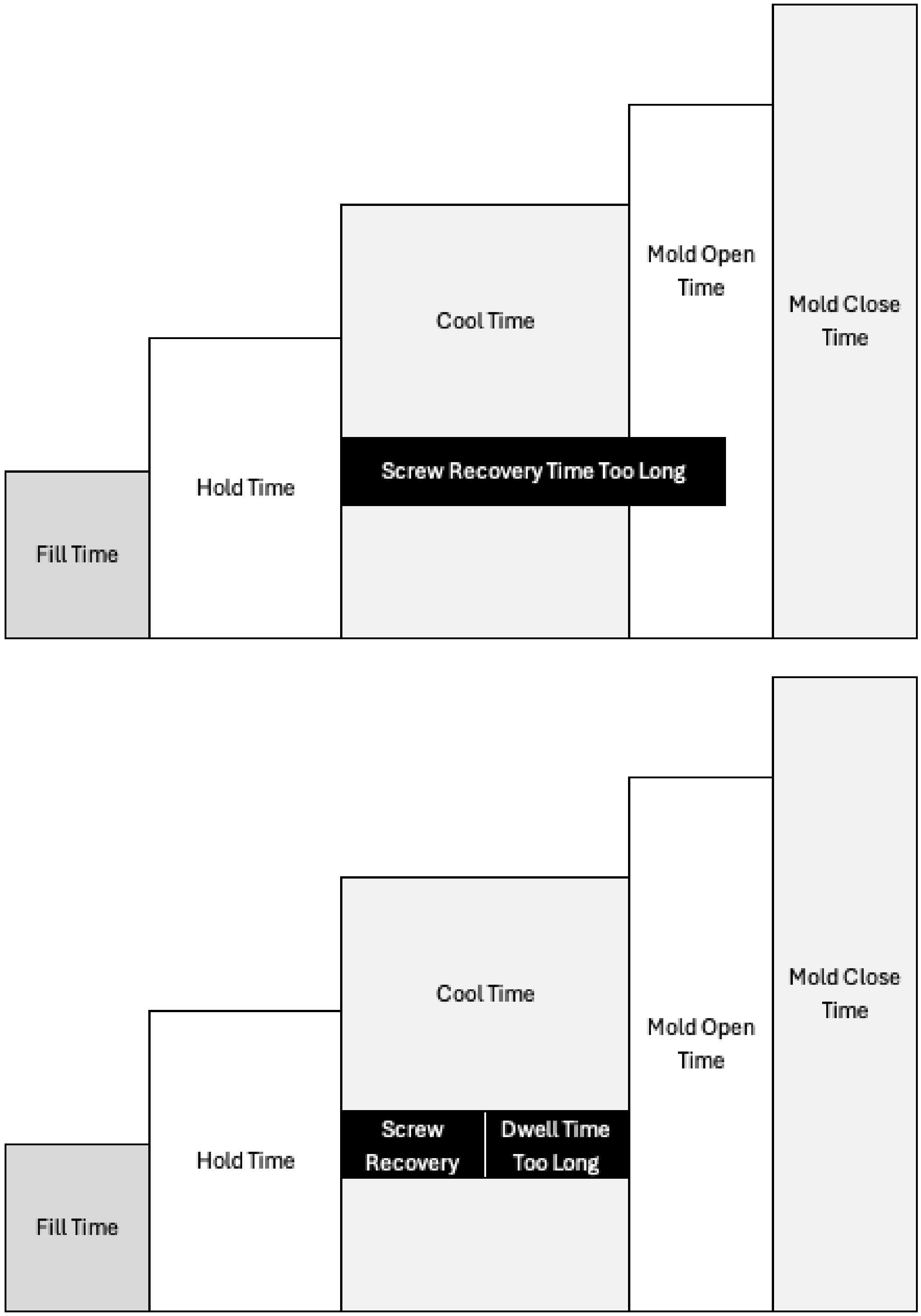

Figure 8 - Time in Seconds

Dwell time is the time the screw spends in the back position waiting to make the next shot. Some dwell time is acceptable, but too much causes the plastic near the barrel walls to absorb excess heat. This makes the melt near the barrel walls hotter than the melt closer to the screw. This causes variations in the cavity fill rate and cooling rate, that affect the part.

If the dwell time is far too long, decrease the screw rotational speed. This will minimize the dwell time. The slower the screw turns, the longer it takes to recover. This causes the screw to mix the plastic while the part is cooling, which results in a more homogenous plastic melt. The screw rotation speed should be set slow enough so the screw does not spend any unnecessary time fully retracted, waiting for the mold to close again.

Repeat and Refine

After the first attempt, the whole cycle time reduction process can be repeated to try to remove a little more time from the cycle. This is normally done after any changes in heats or pressures have been made.

Exercise Four

Cycle Variations

On a machine in your shop, record the dwell time, screw recovery time, back pressures, and screw rpm. If any one of these values changes, notice how it affects the others.

	First Value	New Value
Fill Time		
Screw Recovery Time		
Back Pressure		
Screw RPM		

Instructor Date

Objective Four

Optimal Pressures and Speeds

Once the rough cycle time has been established, it is time to go back and find the limits of the injection pressure window. In-crease the injection pressure until flash starts to form on the parts, or until you are satisfied there is a sizeable gap over the minimum pressure required for a full shot.

An ideal situation is to develop a full shot pressure /temp window. Here, the flash point and short shot pressures are determined for each of three temperatures. One is the lowest practical temperature, the next is the highest practical tempera-ture and the third is a midpoint temperature. Figure 9 is a full shot pressure/temp graph.

It is important not to increase the injection pressure in large increments. A huge increase in injection pressure could send flash behind the slides, stick parts in the mold, or damage delicate sections of the mold. Increase the injection pressure gradually. Record the results in a chart, or use a grease pencil to label each new part with the pressure/temperature setting. The chart can then be made later.

After the parts have cooled to room temperature, measure the key dimensions. Parts almost always continue to shrink as they cool outside the mold.

Allow at least fifteen minutes for the part to cool and stabilize. Larger parts and specialty plastics may take even longer to cool before measurements can be accurately taken.

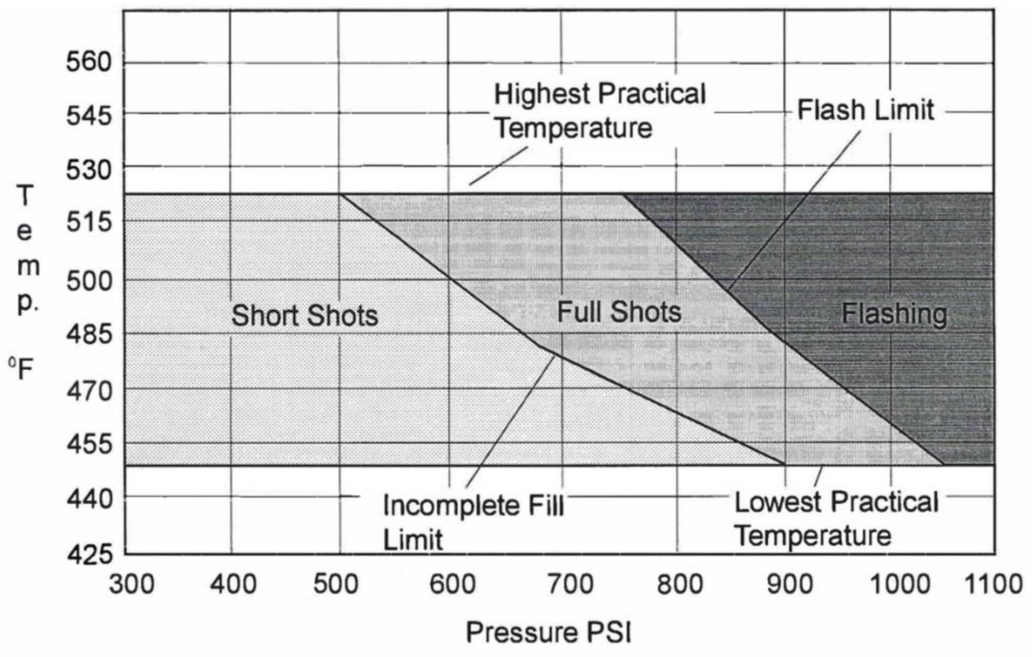

Figure 9 - Pressure/Temp Windows

After the parts have cooled properly, look for the part with the best dimensions. Set the machine to the pressure and/or temperature setting that produced that part. Run the machine for a while at that setting before taking sample parts to the quality control depart-ment for final approval before production.

Injection Speed

Injection speed and injection pressure are somewhat interdepen-dent. A higher injection pressure pushes the plastic with more force, thus increasing the speed.

The general rule when dealing with injection speed is to get the material into the cavity as fast as possible, thereby reducing fill time. Fast fills also tend to minimize orientation in the skin of the parts. The injection speed should be set as high as possible with the established injection pressure without causing any surface defects.

Objective Five

Adjusting Barrel and Mold Temperatures

The viscosity of a plastic material is decreased as the barrel temperature rises. Viscosity is the amount of resistance to flow. Low viscosity means a material is runny, like water. High viscosity means a material is thick and slow flowing, like honey. In other words, the hotter the plastic, the runnier it gets. The cooler the plastic, the thicker it gets.

Changing Temperature Settings

Typically, temperature settings should be changed gradually. It takes time for a temperature change to spread throughout the entire barrel. It is best to wait ten to fifteen minutes after new set points have been reached before expecting the full effect.

Changing the melt viscosity usually means you will also have to make other changes in the process. As in the example used earlier in this lesson, a change in plastic viscosity can require changes in injection pressure, injection speed, or mold temperature to rebalance the process.

Barrel Temperatures

Barrel temperature is controlled with individual electric heater bands wrapped around the barrel. The barrel temperature, along with the back pressure, determines the temperature of the melt. Figure 11 shows examples of temperature controllers that have been set too high and too low. The barrel heats should be adjusted in the following situations:

Raise barrel temperatures if:

1. Material purges out of the nozzle lumpy or corkscrews
2. Screw rotation is labored
3. Parts cannot be filled out with a reasonable injection pressure

Lower barrel temperatures if:

1. Material discoloring
2. Material is degrading
3. Parts are flashing even at low injection pressures

Nozzle Temperature

Nozzle temperature is controlled by electric nozzle heater bands. Nozzle heater bands are connected to PID temperature controllers. Initial nozzle temperatures are normally set to the same temperature as the front barrel zone. If the plastic is freezing off in the nozzle, the nozzle heats can be raised. Likewise, if the material is drooling out of the nozzle, lowering the temperature is one of the cures.

Mold Temperature

If a mold is too hot, the plastic part will take longer than necessary to solidify. Over half of the cycle time of the typical injection molded part is devoted to cooling. In general, cooling parts quickly means better productivity.

Excessively cold molds start to cause problems with the flow of molten plastic. The plastic can start to solidify against the cavity walls before it packs out. This usually shows as surface defects. Cold molds often require such high injection pressures that excess stress is put into the parts.

Usually, the mold temperature should be raised if the cavity is not filling easily, or if the part surface finish is dull or shows flow lines. A lower mold temperature can be experimented with if the cycle time is unacceptably long.

In some cases, it may even be necessary to set a different temperature on each half of the mold. Occasionally, different temperatures are used on the two mold halves if the parts are sticking in the wrong half of the mold, or as a cure for warped parts.

Self-Test

1. For the molding technician, making parts that are dimensionally accurate usually means:
   a. Controlling melt viscosity
   b. Controlling flashing
   c. Controlling shrinkage and packing
2. Why should you keep machine parameters in the middle of their process windows?
   a. To make the process more repeatable
   b. To make more room for error when making changes
   c. So a change in one parameter will not throw the entire process out of tolerance
   d. All of the above
3. Revolutionary changes should be made:
   a. When the process is not making scrap
   b. When the process is under control
   c. On long runs with the same mold
   d. To establish the boundaries of the process window quickly

4. When changing machine settings:
    a. The evolutionary method is generally better
    b. Either the evolutionary or revolutionary method can be used
    c. The revolutionary method is generally better
5. Short shots have more of a chance of damaging the mold than flashing:
    a. True
    b. False
6. The transfer position is generally set until:
    a. The gates freeze
    b. Cavity is 98% full
    c. The sprue is full
    d. The runner is half full
7. In general, you should reduce the hold time until:
    a. Just before the cavity fills
    b. Just after the cavity fills
    c. Just until the gates freeze
    d. Just after the gates freeze
    e. Just before the part is ejected

8. Cooling time is too short if parts are sticking in the mold:
   a. True
   b. False
9. Which of the following will speed up screw recovery?
   a. Decrease back pressure
   b. Increase back pressure
   c. Lower the barrel temperature settings
   d. Decrease the screw RPM
10. One reason to lower the barrel temperature might be that:
   a. Parts are not filling completely even with good injection pressure and speed
   b. Parts are flashing even at low injection pressures
   c. Screw rotation is labored
   d. Plastics is purging out of the nozzle lumpy or in corkscrews

Glossary

Dwell Time - the time the screw spends in the back position, waiting for the next shot.

Evolutionary Method of Change - method of changing machine settings in small increments, carefully measuring and recording changes in the process.

Process Window - a range of acceptable values of a molding parameter.

Revolutionary Method of Change - method of changing machine settings in large increments, quickly finding the outer bounds of the process window.

Sink Marks - depressions or dimples on the surface of the part caused by under-packing.

Viscosity - a material's resistance to flow. Higher viscosity makes the material thicker and less likely to flow (like honey). Lower viscosity makes a material thinner and runnier (like water).