Molding Materials and Process Troubleshooting

LESSON 3: Problem Solving Strategies

Lesson 3: Problem Solving Strategies

Good troubleshooting and process control start with a scientific outlook. One of the best ways to analyze the process is to look at injection molding from the plastic's perspective. Ask yourself what forces are acting on the molecules. What happens to the plastic as it moves from the barrel to the mold?

Solving problems is often a matter of puzzling though the effects of shearing, compression, orientation, cooling rate, ejection force, and even how fast air escapes the cavity. Understanding the process as a whole will make troubleshooting faster and easier. Careful analysis almost always yields faster results than trial and error, or random knob turning.

In this lesson we will discuss some common sources of molding problems, and some techniques to help solve these problems quickly.

Objectives of Lesson 3

1. Learn the objectives of troubleshooting
2. Learn about the sources of molding problems
3. Learn approaches to problem solving
4. Learn about establishing process windows

Objective One

Troubleshooting Objectives

Minimize Troubleshooting Time

Given unlimited time and resources, almost any molding problem can be solved. Profitable molding comes from solving problems quickly. When part defects or other problems arise, your goal is to solve the problem as fast as possible. Every minute that a defect or problem goes unsolved, the machine is adding up downtime, or making scrap. Solving problems quickly means approaching the process logically, with a good understanding of how the machine, the mold, and the plastic material interact.

Anticipate Changes in Quality

A quality part is a part that exceeds certain specified standards of size, visual appearance, strength, and other specifications. When a part dimension or physical property changes, molding techni-cians must know the process well enough to anticipate changes. A good technician can head off poor quality parts by changing the process before the parts fall below specifications. You need to monitor the quality of your parts, and know the process well enough to select control variables that will bring the part back into specification.

Set Reliable Process Conditions

Solving problems quickly is only half the battle. The other half is keeping problems from recurring. As a molding technician, one of your goals is to know the process well enough to have confidence in your solutions. Having confidence means knowing how much time to give the machine or mold to "settle" a change into the process. Take the time to establish process windows during setup and troubleshooting. Whenever possible, establish your optimum parameters towards the middle of the process window.

A process window is a range of acceptable values. All other things being equal, if a machine seems to make good parts with injection pressure between 200 and 900 psi, try to run toward the middle of the acceptable range.

Keeping parameters in the middle of their process windows gives you more room for error, and more room to adjust the overall process. Changes will be faster and easier to make. The process windows of the important parameters (like injection pressure, injection speed, mold temperature, barrel temperature, etc.) are interrelated.

Keeping to the middle of each parameter's process window means changes will be less likely to affect the other parameters. If you run an important parameter towards the edge of its range of acceptable values, ordinary machine fluctuations could throw the process out of tolerance, and bad parts could be made.

Objective Two

Sources of Molding Problems

Problems with the Machine

Molding problems have many causes. In this section, we will look at problems with the machine, problems with the mold, and problems with the melt.

Injection molding machines are complex devices. They require constant care and attention to keep them running smoothly and consistently. Even a properly maintained machine is subject to ordinary wear. The following is a discussion of some of the most common problems that start with the machine .

Worn Screw Check Ring

The screw check ring (non-return valve) slides back and forth with the screw. When the screw starts to inject plastic, the ring slides back and closes off the rest of the barrel. This seals off the barrel, allowing the screw head to keep pressure on the melt. If the check ring is worn or damaged, melt can flow back in to the screw flights, as shown in Figure 1. The screw will not be able to keep pressure on the melt. When this happens, the part is usually under-packed.

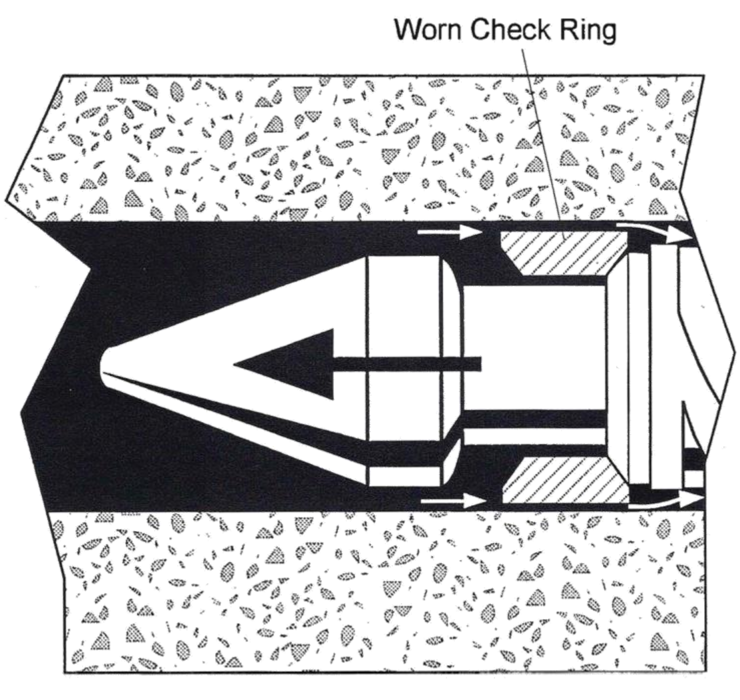

Figure 1 - Worn Check Ring

Worn Screw

As the check ring, screw, and barrel wear (Figure 2), more and more plastic can slip over the flights of the screw. The reduces the pumping efficiency, and increases the screw recovery time.

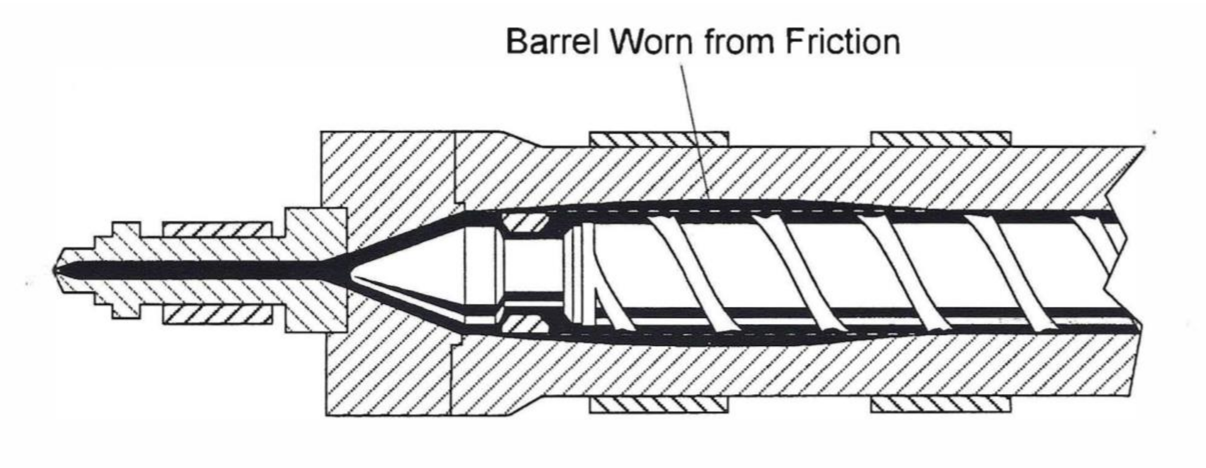

Figure 2 - Worn Screw and Barrel

If you notice a problem with the cavity, the screw recovery, or with the ability of the screw to hold a cushion, make a note of it. The maintenance department can check the screw and barrel the next time the machine is down.

Exercise One

Mold Maintenance

Ask your shift supervisor, or maintenance supervisor, about the server, check ring, and barrel condition on some of your presses. Record whether they require, or have recently undergone, repair.

Mold Number	Check Ring Condition	Barrel Condition	Recent/Needs Repair

Instructor Date

Hydraulics

The hydraulic system delivers the power to move the screw forward during injection, and backwards during recovery. Changes in the hydraulic system often show up as changes in the injection pressure or speed. Keeping the hydraulics smooth and trouble free is an important part of problem-solving.

Hydraulic Oil Temperature

Oil is a viscous liquid. Its viscosity, or resistance to flow, decreases as the temperature increases. Hotter oil flows faster than colder oil. Oil temperature determines how much oil pressure comes to bear on the cylinder. If the oil temperature fluctuates, the injection pressure will also vary. Figure 3 shows a hydraulic oil temperature gauge reading that is too high.

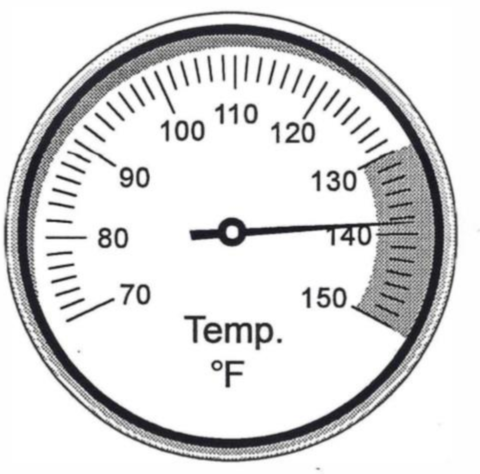

Figure 3 - Hydraulic Oil Temperature Guage

Hydraulic Valves Sticking

When hydraulic valves stick, they produce a general change in the overall injection force. If the first stage hydraulic valve sticks, there will be a momentary drop in the pressure on the screw. First stage injection pressure and speed are likely to vary dramatically. This will cause variations in the cavity fill rate or packing pressure.

If the hold pressure, hydraulic pump sticks, the hold pressure will drop. If the holding pressure drops below the process window, or range of acceptable values, the parts will be bad.

Heater Bands

Heater bands deliver the heat to the barrel to bring the plastic to the correct melt temperature. Good melt temperature is critical to making quality parts. If a heater band is loose, or malfunctions, it will not deliver the same amount of consistent heat to the barrel.

Surrounding heater bands can overheat trying to compensate for the damaged band, as illustrated in Figure 4. When this happens, the overall heat profile changes. The plastic can actually be too hot in some areas, and too cold in others.

Overheated plastic will lose viscosity. It will flow into the cavity faster. Cooler melt will be more resistant to flow. Heater bands must be well attached, and carefully monitored, to maintain uniform correct melt temperature.

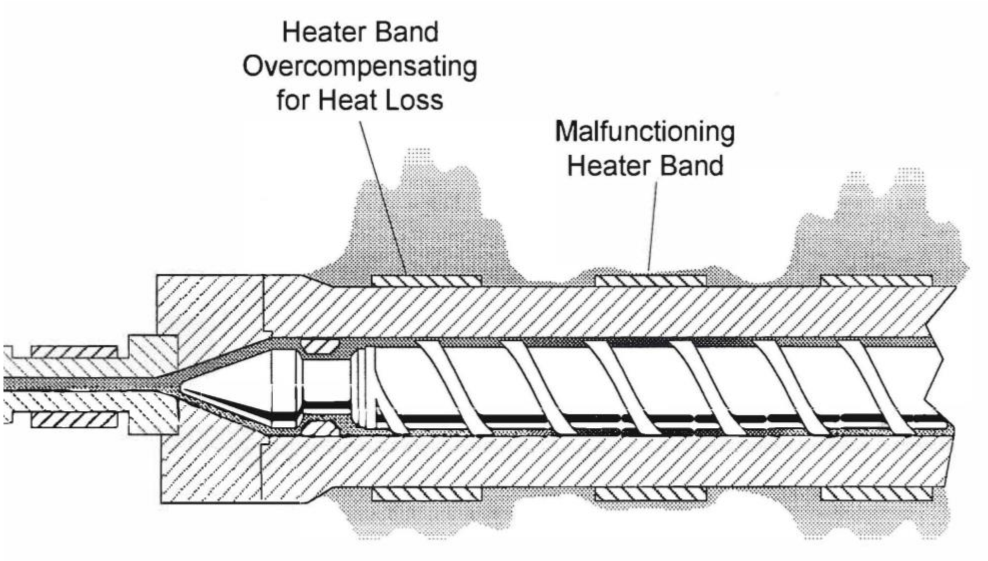

Figure 4 - Damaged Heater Band

Malfunctioning Encoders

Encoders send signals to the machine controls based on the position of the screw or clamp. One of the most important encoders is the one that controls the positional change from injection velocity to hold pressure. If this sensor is defective, the process will not make the transition to hold pressure at the correct time. Getting the holding pressure right is critical to proper packing of the part.

The mold heating and cooling depends on having good flow of heating and cooling fluids through the channels. When the heating and cooling lines are blocked by rust or other buildups, fluids cannot flow through the heating and cooling channels in the mold, as illustrated in Figure 5.

Sometimes a blockage in one channel is hard to notice. Without fluids, one section of the mold changes temperature. Uneven mold temperature will probably cause sections of the part cool at different rates. This can cause warpage, shrinkage, and many other problems.

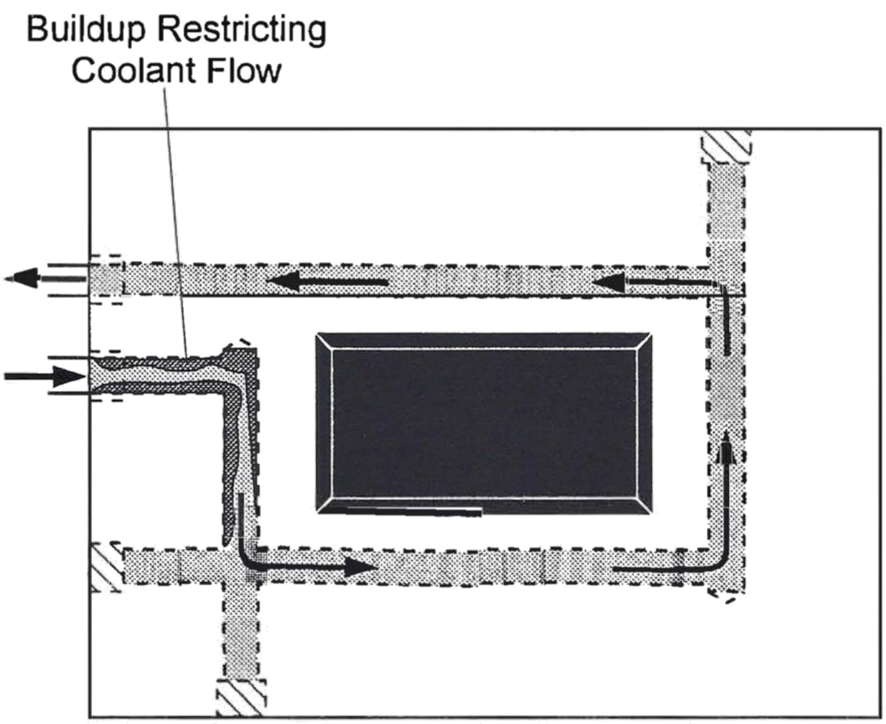

Figure 5 - Blockage in Cooling Channels

Problems with the Mold

Design Deficiencies

Many molding problems stem from the design of the mold. Some problems result from poor mold design, while others are necessary evils caused by parts that are inherently difficult to mold.

As a molding technician, you may not have a chance to influence the design of the mold. You can, however, try to understand the molding process well enough to compensate for design problems.

Easy To Change Mold Problems

Some mold features are changeable. Generally, you should consider making these kinds of changes only after trying changes in the process variables.

The following mold changes generally do not take too much time or money. Most of these changes can be made without redesigning the mold.

Vents

Many part defects stem from air trapped in the mold. If air in the cavity does not escape properly, it can turn to steam, or burn, causing serious part defects. You can help the air escape by changing the vents.

Vents are tiny channels at the end of the flow path. These channels allow air trapped in the cavity to leave the mold. Often, the size and location of the vents are changeable. You can increase the size of the vents, as shown in Figure 6, or add more vents at a particular spot, to improve removal of air.

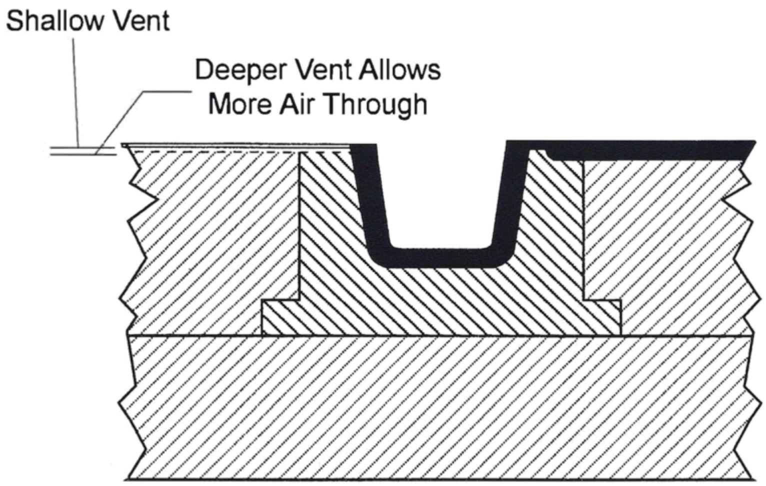

Figure 6 - Vents

Ejector Pins

Poor ejection can cause dimensional problems. If the ejector pins are too long or too short, they can leave an unacceptable hole or boss on the part. Changing the length of the pins is a fairly simple procedure. Although time consuming, you can change the length and size of the ejector pins to improve ejection.

Exercise Two

Vents

Locate an open mold and examine the number, size and location of its vents. Can the vents be changed easily? Make sure the mold is locked out beforehand.

Mold Number			
Part Produced			
Number of Vents			
Size of Vents			
Location of Vents			
Can they be easily changed?			

Instructor Date

Gate Size

Small gates can restrict the flow of plastic into the cavity (Figure7). Often the gates are designed and built smaller than the optimum size. Small gates can shear the plastic as it passes through. Shearing at the gates can cause orientation. As the chains of plastic are stretched out and strung through the narrow gate, they tend to line up (orient).

If the gates are too small, the plastic will be oriented too much. If the part does not have time to cool and return to normal, the orientation will be molded into the part causing molded-in stress. Gates can usually be enlarged in a company's in-house tool shop.

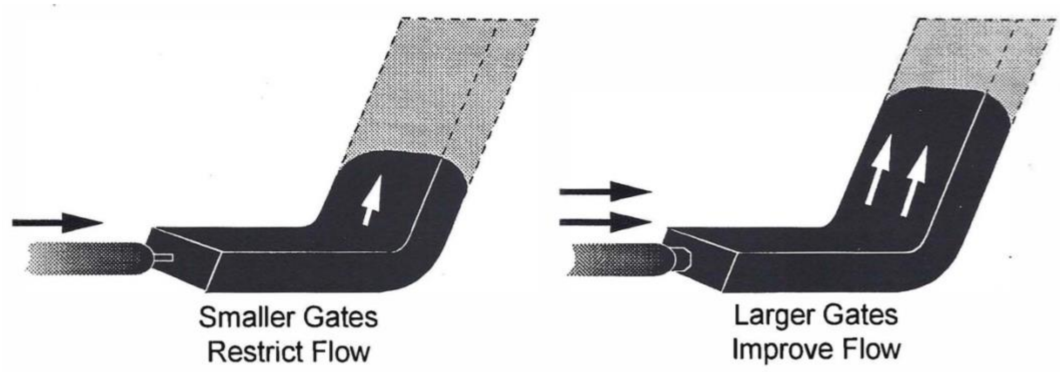

Figure 7 - Gate Size

Damage

Poor quality parts may result from damage to the mold or the cavity surface. Nicks and damage on the cavity surface may cause imperfections on the part. Nicks or scratches can also make tiny undercuts that stick the part in the mold. Most scratches and nicks can be repaired. They can be filed out, and then repolished.

Hard To Change Mold Problems

Molding technicians usually cannot change the following mold problems.

Unbalanced Flow Path

An unbalanced flow path creates a different fill rate for different sections of the part. Some sections of the part may end up overpacked, while other sections are under-packed. You may have to experiment with melt temperature and injection pressures to find a fill rate that makes the best part possible.

Thin Walls or Uneven Wall Thickness

Parts with very thin walls can present many problems. It is sometimes difficult to fill thin walled parts evenly and completely before the part solidifies. For these types of parts, usually you must experiment with higher melt temperatures and mold temperatures. Higher melt and mold temperatures will decrease the melt viscosity, making it flow faster and easier into the cavity. Parts designed with uneven walls often have fill problems. The thin walled sections are harder to fill than the thick walled sections. When a thin walls fill, they begin to cool, solidify, and shrink before thick walls do. Uneven cooling and filling can lead to many dimensional problems. You may have to select an injection speed or barrel temperature that is the best compromise.

Exercise Three

Mold Problems

Locate one of the hardest to fill molds in your shop. What problem do you think is responsible for this mold being difficult to fill?

- Small Gates .... ☐
- Thin Walls .... ☐
- Hard Flow Plastics .... ☐
- Cold Mold .... ☐
- Hydraulic Core Pull Lines .... ☐
- Long Flow Path .... ☐

Instructor Date

Shallow Draft Angles

Some ejection problems stem from a shallow draft angle. Draft is the angle or slope given to the walls of a part. Sloping the walls of the part makes it easier to eject. If the draft angle is very steep, the part may not eject well. In this case, you may have to experiment with the ejection force and mold open time to get the part out of the mold as cleanly as possible.

Built in Undercuts

Some parts are designed with undercuts that cannot be moved or changed. These undercuts can make it very difficult to eject the part. You may have to experiment with cooling time and mold open time to eject the part before it over-solidifies. If a part with a built in undercut cools and shrinks too much, it may stick in the mold.

Problems with the Melt

Different Material

Some molding problems can stem from variations in the melt properties. Plastic material from two different suppliers may have the same melt index and other specifications but actually mold quite differently. Plastic material can even change from one manufactured lot to another. Even if all your material is from the same supplier, minor variations from one lot of plastic to the next can change the process. Different lots of material might have slightly different molecular weights. Pay attention to changes in the supply of material. Expect process changes whenever changing material.

Additive and Filler Problems

Additives and fillers can separate from the base material. This usually happens because of over-heating or excess time in the barrel. Separated fillers can build up on the cavity walls, causing dull spots in the part.

Moisture and Drying Problems

Many plastics are hygroscopic materials. Hygroscopic means they tend to absorb water, even from the air. If moisture makes its way into the cavity, it can cause splay, bubbles, and other part defects. Most plastics need to be properly dried before molding.

Problems with the Operator

Inconsistent cycling

On semi-automatic machines, the operator actually determines the overall cycle time by how fast they perform certain tasks (Figure 8). The mold open time depends on how fast they open and close the safety doors, or how fast they reach in and pull out the molded part. Operators can get tired, lengthening the overall cycle time. As the cycle time gets longer, the material must wait longer in the hot barrel. The melt temperature rises causing changes in the plastic's flow rate. The fill rate and packing of the part can change because of minor changes in the cycle time.

Not Enough Cooling Time

In many shops, the operator will have other things to do while the machine cycles. They may have to sort or trim parts, insert pieces, take measurements, or do many other things. Also, if you stack parts in a bin, a conveyor, or box before they are cool, they can stick together. Pulling them apart often damages the part surface, or causes warping. Allow plenty of cooling time before stacking parts.

Figure 8 - Operator Removing Parts from a Mold

Objective Three

Problem Solving Approach

Many times you can solve molding problems quickly and easily simply by having a logical and consistent approach to problems. The following system, as illustrated in Figure 9, can help you look at molding problems consistently.

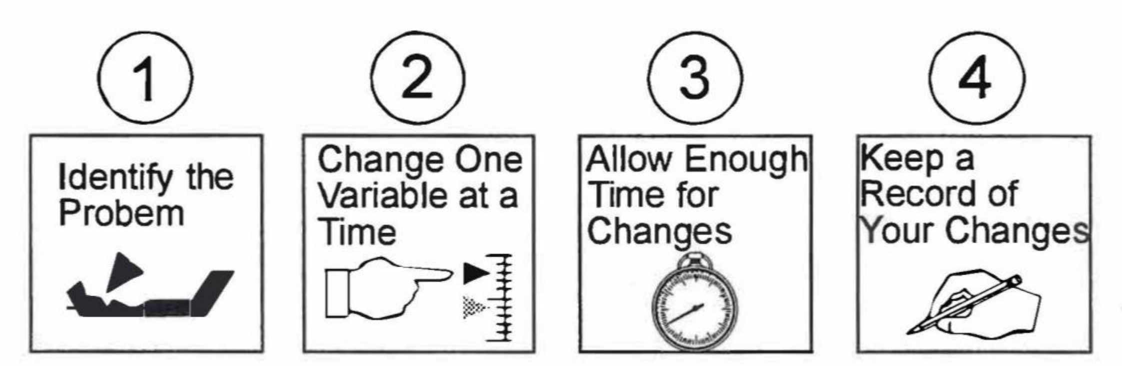

Figure 9 - Problem Solving Approach

Identify the Defect or Problem

The first step is to define the problem clearly. Is the problem consistent with every part, or does it come and go? Is the defect in the same spot every time? Does it occur in the same cavity every shot? Clearly defining the problem will lead you to the solution faster.

For example, splay or silver streaking in the part could be due to

moisture in the melt, excessive injection speed, or lack of packing.

Choose the potential solution that is easiest to prove or disprove.

In this case, try varying the speed or pressure. If that does not work, then try changing the material in the hopper to see if it was wet.

Change One Variable at a Time

It is a good idea to pursue one solution at a time. Most of the process variables are interdependent. Changing one variable often changes the range of acceptable values for other variables.

For example, changing the barrel temperature may change the range of acceptable injection speed settings. When making changes, change one variable at a time. If a change does not solve the problem, return the variable to its original setting before trying something else. Changing too many conditions at one time can cause a lot of confusion. It is easy for the process to go completely out of control with too many changes.

This does not mean, however, that you should never change more than one variable at a time. Often one change must be made to balance the effects of another change. For example, if you raise the barrel temperatures to improve the surface texture of the part, you may have to also lower the boost pressure to keep the part from over-packing.

Allow Enough Time for Changes

Be sure to allow time for your changes to take effect. Many of the molding variables are interrelated. It takes time for the overall process to achieve an equilibrium. Changes to pressure or speed settings may take five or six shots to stabilize. Changes in temperature settings can take twenty or thirty cycles to stabilize.

Keep a Record of Your Changes

Record new information every time you make a change to the process. You should also record any special notations so that the following shifts can learn from your solutions. It is important to keep records at the machine to help other operators. It is also important to keep a permanent record of the molding process on the setup sheet. Ask your supervisor how your shop records changes in the process.

Exercise Four

Find a job in which recent changes have been made. Check the shift parameter sheet or log sheet. Note the changes that have occurred over the last week, and who made the entry on the sheet.

Change	Who Made the Entry

Instructor Date

Objective Four

Establishing Processing Windows

When a mold is set up in a machine for the first time, the molding conditions are set, and a full part is established. This process by itself does not ensure that the molding conditions are the best that they could be. For example: if the processing conditions are set so the cavity just barely fills, a small variation in the process could cause the parts to be short.

Likewise, the mold might run just as well with a 30° lower barrel temperature. Generally, a lower barrel temperature reduces cycle time because it reduces the cooling time.

You should take some extra time during start up or sampling to establish a processing window. The time it takes to complete this controlled experimentation will normally pay off in the long run.

There are a few different types of processing windows that can be used. The simplest types use only two different molding variables, and result in an easy to read molding area diagram. The diagram establishes the processing boundaries that generate an acceptable part. These boundaries are often referred to as the processing window.

Barrel Temperature versus Pressure Diagram

One of the most important molding area diagrams shows the relationship between barrel temperature and injection pressure.

Here, the lowest and highest practical barrel temperatures are determined and used for two of the test temperatures. A third temperature may be selected that is halfway between the two extremes. It is usually faster to start with the lower temperature, and then move to the higher ones. This is because, usually, the barrel can heat up faster than it can cool down.

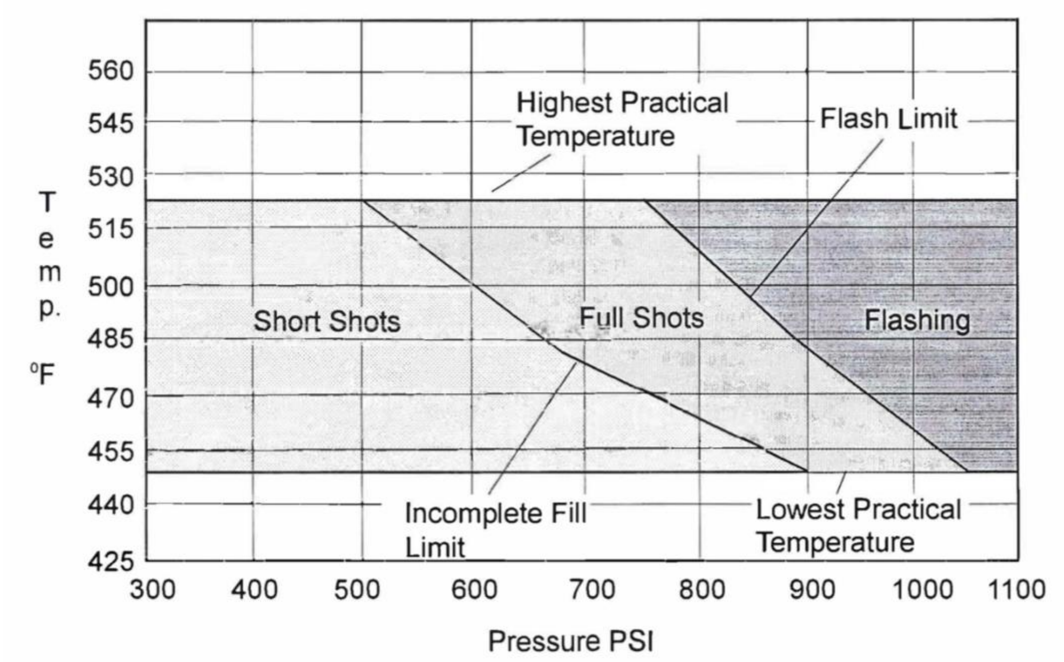

Figure 10 - Barrel Temperature vs. Pressure Diagram

Parts are shot at each temperature. The pressure is varied so that both the short shot and flash points may be determined. Six points are established. When plotted, these show the processing window for a full shot. Generally, a temperature in the lower half of the chart will be selected for production usage. A lower temperature usually means in a faster cycle time. An exception is thin walled parts. If they are molded at too low a temperature, they will be highly oriented. If the parts fill, but come out slightly warped, select a higher temperature.

Remember to allow the barrel to heat soak long enough between each temperature change. When setting the injection pressures, it is recommended that injection velocity and hold pressures are the same so that the procedure is simplified.

Mold Temperature versus Pressure Diagram

Another important molding area diagram shows the relationship between mold temperature and injection pressure. The procedure is similar to the one just discussed. Here, the barrel temperature will be held constant, and the mold temperature varied. A barrel temperature is usually selected from the middle to lower range of the previous

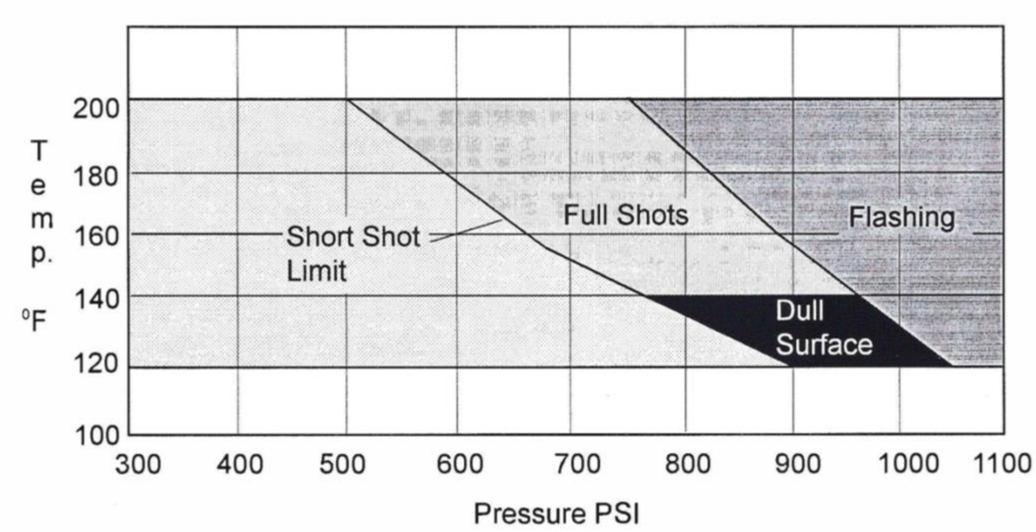

Figure 11 - Mold Temperature vs. Pressure Diagram

diagram. Often, more than three mold temperatures need to be tried to find the optimal range. Typically, the lower mold temperatures will allow a full shot to be made, but will produce an unacceptable surface on the part. Final mold temperature for production should be something in the range of 20° to 30°F warmer than this.

Dimensional Analysis

Once the optimal barrel and mold temperatures have been established, a dimensional analysis can be completed. Accurate dimensional analysis depends on quick, consistent cycle times. If the overall cycle time is too long, or varies from shot to shot, part cooling time will be too long. Long cooling times often make large parts that would ruin the dimensional analysis. The cycle time should also have been minimized by this point. The cycle time needs to be constant, since excess cooling causes the parts to be larger and would affect the results.

A key part dimension, such as part length, width, or diameter, should be selected from the part drawing. The tolerance band also needs to be noted. The barrel and mold temperatures should be set and allowed to heat soak. Then, a series of different injection pressures can be tried. For this purpose, 20 to 50 PSI increments are usually adequate.

Mold a part at each injection pressure and label it with a grease pencil. Lay the parts out on the bench in a manner that will allow them to cool evenly. After they have cooled, measure them with a dial caliper or micrometer. Plot the

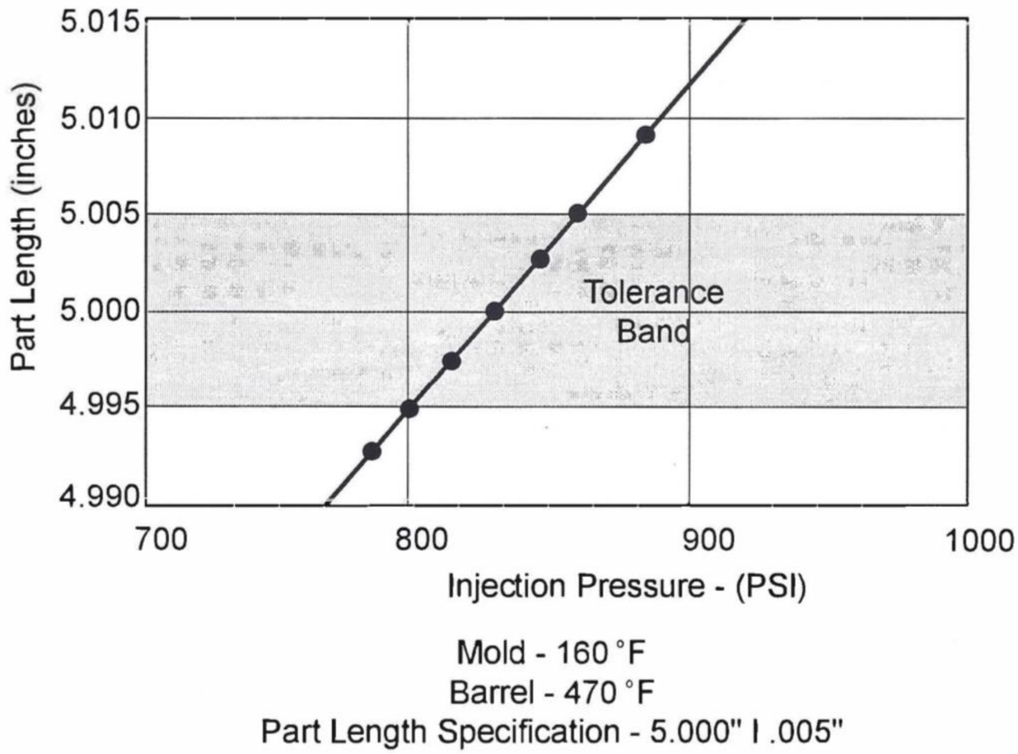

Figure 12 - Dimensional Analysis

points on a simple graph. Select the parts from the middle of the acceptable range and send them to the quality control department for complete dimensional and physical checking.

This set of procedures should ensure that the process has been optimized and that the parts are being produced well within the limits of the various processing windows. This will establish a repeatable and reliable process.

Self-Test

1. A worn screw check ring is likely to make parts that:
   a. Are over-packed
   b. Are under-packed
   c. Cause flash
2. Blocked heating or cooling channels are most likely to cause:
   a. Over-packing
   b. Splay
   c. Warpage, excess shrinkage, or distortion
   d. Bubbles
3. Shearing at the gates often causes excess orientation which can lead to:
   a. Bubbles
   b. Molded in stress
   c. Crystallinity
   d. Excess Viscosity

4. Keeping machine parameters in the middle of their process windows:
    a. Makes the process more repeatable
    b. Allows more room for error
    c. Allows for normal machine fluctuations
    d. All of the above
    e. None of the above
5. A worn screw will often:
    a. Decrease the recovery time
    b. Increase the recovery time
    c. Increase the dwell time
    d. Have no effect on recovery time
6. One possible solution for parts with thin walls that do not fill properly is to:
    a. Increase the melt temperature to decrease viscosity
    b. Decrease the melt temperature to decrease the viscosity
    c. Increase the melt temperature to increase viscosity
    d. Decrease the melt temperature to increase viscosity

7. Longer cycle times can mean the plastic spends too much time in the barrel. This could cause:
   a. Under-packing
   b. Over-packing
   c. Short shots
   d. Lower melt temperature
8. A sticking hydraulic valve will most likely cause:
   a. Over-packing
   b. Flashing
   c. Burn marks
   d. Molding variations
9. When troubleshooting, it is a good ideas to change only one machine parameter at a time because:
   a. Most of the process variables are interdependent
   b. Most of the process variables require complete attention
   c. Most of the process variables are independent
   d. None of the above

10. Which two variations used to establish a processing window?
    a. The Barrel Temperature vs. Viscosity diagram
    b. The Barrel Temperature vs. Injection Pressure diagram
    c. The Mold Temperature vs. Barrel Temperature diagram
    d. The Mold Temperature vs. Injection Pressure diagram
    e. The Barrel Temperature vs. Cavity Pressure map

Glossary

Draft - the angle or slope given to part walls to aid in ejection.

Hygroscopic - a material's tendency to absorb water from the atmosphere.

Check Ring - device near the tip of the screw that closes off the barrel during injection, preventing plastic from flowing back into the screw flights.

Vents - thin channels at the end of the flow path that allow air to escape the cavity.