Molding Materials and Process Troubleshooting

LESSON 4: Packing and Fill Problems

Lesson 4: Packing and Fill Problems

Injection molded parts are formed by injecting molten plastic, at high speed and pressure, into a mold, then holding the plastic in the cavity until the part solidifies. Getting the right amount of plastic into the cavity is called packing the part. Pushing too much material into the cavity is called over-packing the part. Packing the cavity with the right amount of plastic is critical to good part quality.

In this lesson we will look at some of the more common part defects related to filling and packing the part.

Objectives of Lesson 4

1. Learn how packing and filling affect part quality.
2. Learn the causes of flashing and short shots.
3. Learn the likely causes of sinks and voids.
4. Learn about weld lines.
5. Learn what to do about jetting.
6. Learn why parts stick.

Objective One

Packing and Filling Defects

Packing defects occur when the wrong amount of material is injected into the part. Fill defects arise from problems with the injection speed or melt temperature.

Defects Caused by the Process

Most of the time, filling and packing defects are the result of problems in the process. Filling and packing defects can be caused by machine malfunctions, but most of the time they are caused by incorrect settings in the process. One or more of the process variables is out of tolerance. The out of tolerance variable is not always obvious. Many of the process variables can affect the fill rate. Packing and filling the part correctly depends on injection speed, injection pressure, back pressure setting, mold temperature, barrel temperature, screw recovery rate, and even overall cycle time.

Defects Caused by the Mold Design

Many fill defects come from poor mold design. The channels and grooves in the mold make up the runners through which the melt must travel before it can fill the cavity. The shape and size of this runner system will, naturally, have a great effect on how well the plastic flows into the cavity .

The Melt Front

As plastic flows into the mold, it maintains a melt front. The melt front is the leading edge of the molten plastic. The melt front is slightly cooler than the rest of the plastic behind it. The goal of scientific flow analysis is to keep the melt front uniform, as shown in Figure 1. If the front breaks up or otherwise disrupts the flow of plastic, fill defects appear. Engineers try to design molds that will keep the melt front from breaking up before it reaches the cavity.

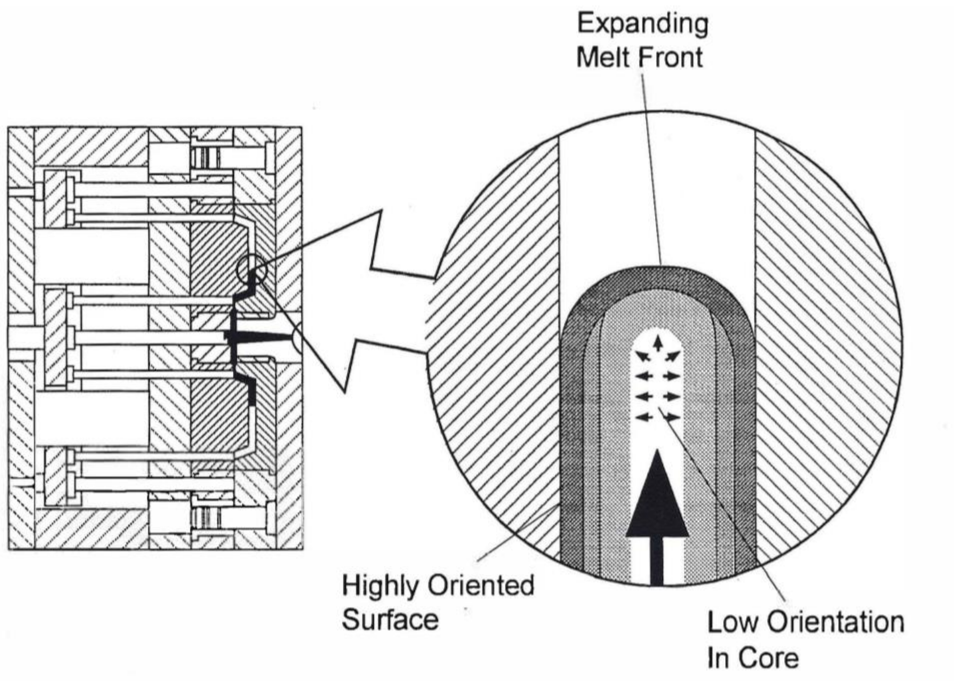

Figure 1 - Melt Flow in Cavity

Mold Temperature

Mold temperature can also have an effect on the packing and filling of the part. The temperature of the mold determines how long the plastic will remain molten inside the cavity. If the plastic stays molten too long, it could over-pack, causing sticking in the mold. If the plastic cools too quickly, the gates may freeze too soon, resulting in under-packing. The part will probably shrink too much as it cools. Good mold temperature is also necessary for proper filling.

Defects Caused by the Plastic Material

Packing and fill defects could also be due to the plastic material itself. Even the best process settings and mold design must work with the melt being used. If you use a melt with a very runny flow rate, you may get flashing. Likewise, contaminated melt, or melt that was not dried properly, may still contain water or impurities to that could be trapped in the molded part. Material defects could be caused by material that is not dry enough, problems with regrind or contamination, using the wrong lot or type of material, or even problems with additives and lubricants.

Objective Two

Flashing and Short Shots

Flash is extra plastic material forced out of the mold. It usually leaves the mold at the parting line, and sticks to the part or to the surface of the mold. Flash escapes the mold either because the forces injecting the melt are too strong, or the forces keeping the mold closed are too weak. Figure 2 shows a part that has flashed at the parting line.

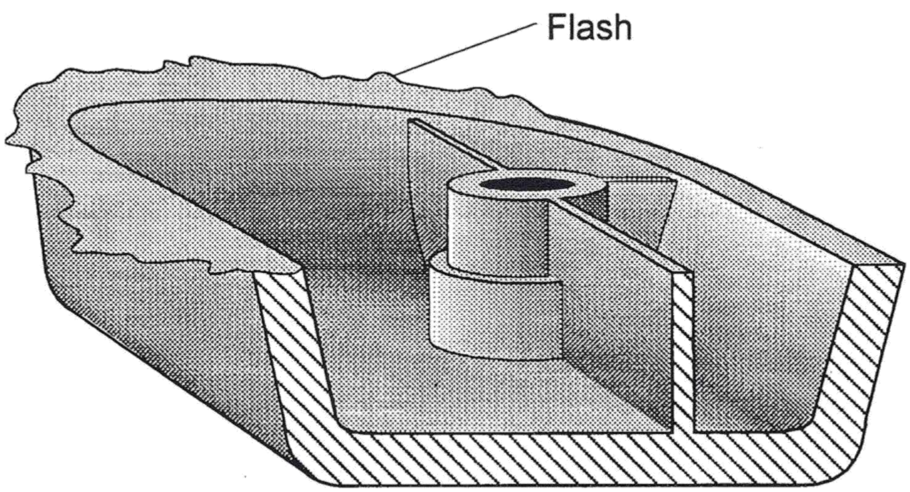

Figure 2 - Parting Line Flash

Injection Pressure and Speed

The most likely cause of flash is too much injection pressure or speed. If injection pressure is too high, the melt will be injected with more force than the mold can contain.

Temperature and Viscosity

Viscosity is the melt's resistance to flow. The lower the viscosity, the easier the melt will flow into the cavity. Even with the correct injection pressure, if the melt gets runnier, it will flow faster into the mold and possibly flash.

Generally, when the melt temperature increases, the plastic gets runnier. There are several reasons why the melt temperature might be too high. The barrel temperature may be set too high. The time the melt spends in the barrel waiting for injection may be too high. The melt can spend too much time in the barrel because the screw recovery time is set too slow, or the overall cycle time is not as efficient as it could be.

Clamping Forces and Mold Seal

The other possible cause of flash is that the forces containing the plastic in the cavity are not sufficient. If the clamp pressure is too low, there will not be enough force to contain the pressure of injection. Even with adequate clamping forces, if the seal at the parting line is not good, flash will occur. The important seal is the area about one inch around the outside of the part. This area is called the land. The land may not seal properly because of flash on the face of the mold, or nicks and gauges on the surface of the mold.

Exercise One

Flash

Find several parts with an unacceptable amount of flash on them. Determine which of the fallowing things caused the flash.

Part Number	Damaged Parting Line	Lack of Clamp Force	Excessive Injection Pressure

Instructor Date

Short Shots

Short shots are the opposite of flash, not enough material is entering the cavity to fill the part properly. Short shots can occur when there is not enough melt, or the forces injecting the melt are not strong enough to get it all into the cavity before it solidifies. Figure 3 shows two examples of short shots, one smaller, and therefore more extreme, than the other.

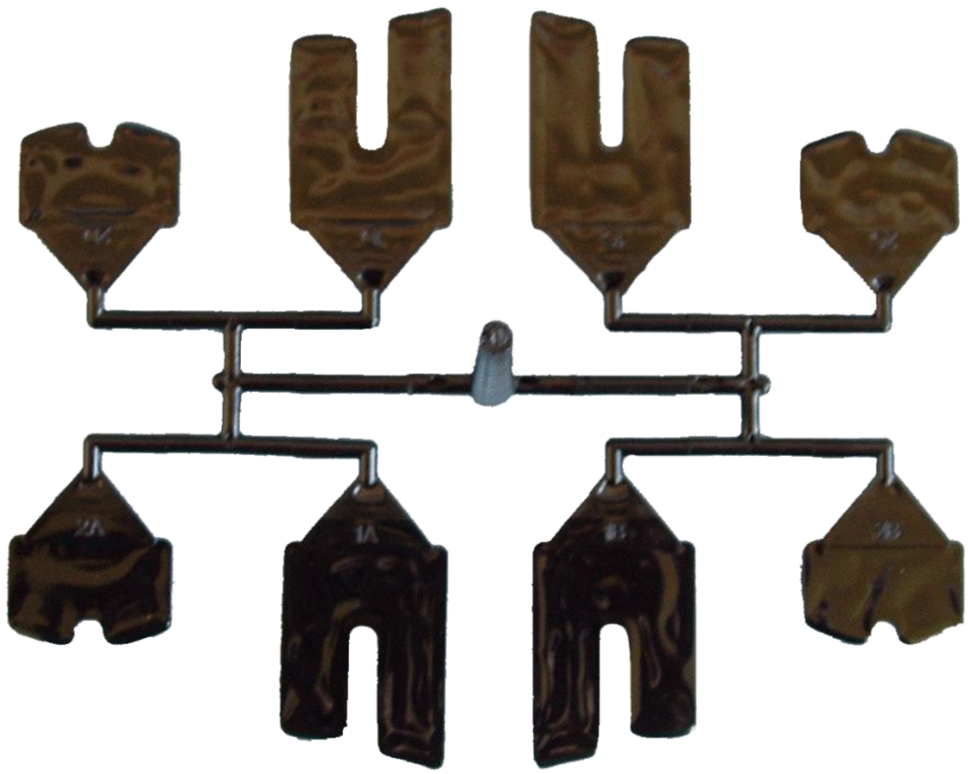

Figure 3 - Short Shots

Shot Size and Injection Pressure

The most common cause of short shots is a small shot size. There simply is not enough material in front of the screw to make the part. Fortunately this is easy to correct. Move the shot size limit switch back to increase the shot size.

If the supply of melt is good, the forces injecting the melt may be too weak. If the injection pressure or injection speeds are too low, the melt will not be injected into the cavity fast enough to fill the part properly.

Temperature and Viscosity

Viscosity is the melt's resistance to flow. The higher the viscosity, the more force required to push the melt into the cavity. Even with a high injection pressure, if the melt is too thick and cool, it will not flow well enough to fill the part.

Generally when the melt temperature decreases, the plastic gets thicker. There are several reasons why the melt temperature might be too low. The barrel temperature may be set too low. The time the melt spends in the barrel waiting for injection may be too short. Then the melt does not spend enough time in the barrel, the screw does not have a chance to mix and heat the plastic properly. Time in the barrel might be too short because the screw recovery time is set too fast, the screw rpm is set too fast, or the back pressure setting is too low.

Exercise Two

Short Shots

Locate a short shot part that is almost filled. Circle or mark the unfilled area. Which area of the mold failed to fill.

Part Number	Thin Rib or Projection	Opposite Gate	Area that Did Not Back Fill

Instructor Date

Other Causes of Short Shots

On the machine, there are a number of other settings that can cause short shots. If the mold temperature is too cool, the plastic will solidify too quickly as it enters the mold. This could cause the gates to freeze before the melt fills the cavity.

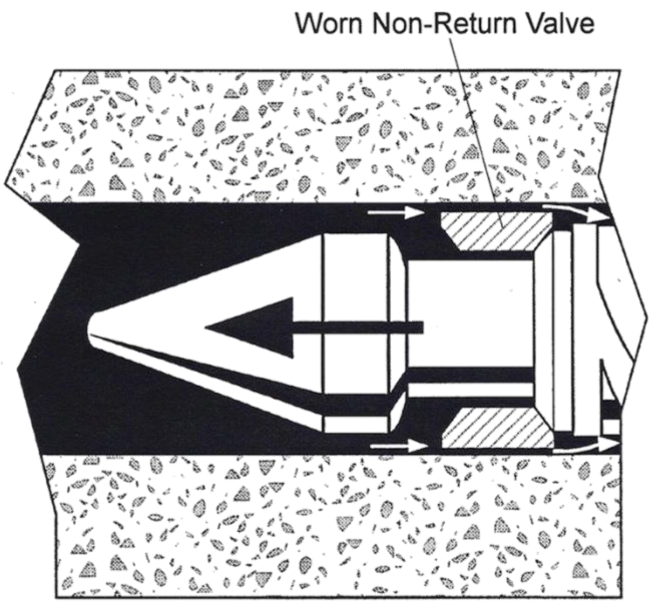

Figure 4 - Backflow During Injection

The Check Ring (non-return valve) slides back during injection, keeping plastic ahead of the screw tip. If the non-return valve is worn and does not seal against the walls of the barrel, plastic will flow over the ring and back down the screw flights, instead of flowing into the part. Figure 4 shows back flow over a worn check ring during injection.

Mold Filling Window

Most machines have a window where parts will fill correctly. A window is a range of acceptable settings for a process parameter. The fill window on most machines is determined by injection pressure, and melt or barrel temperature. Too much pressure, and the melt will be forced so hard into the mold that it flashes. Too little pressure, and you will make a short shot. If the barrel temperature is too high, the melt will get runny and tend to flash. If the barrel or melt temperature is too low, the melt will become thicker and more resistant to flow. You will tend to make short shots. Making good parts means finding the combination of pressures and temperatures that will put you comfortably in the fill window. Figure 5 shows an example of the relationship between pressure, temperature and fill conditions.

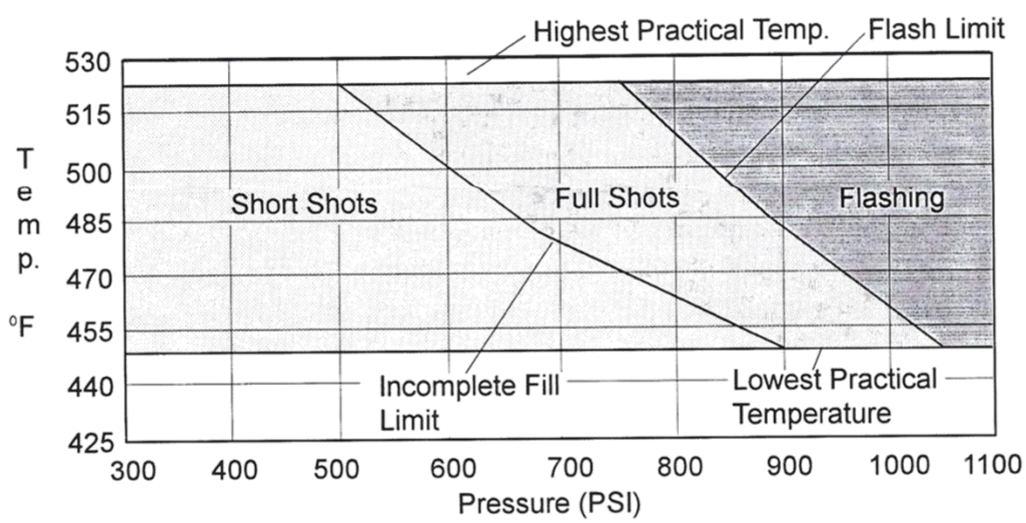

Figure 5 - Good Pressure and Temperature Combinations

Objective Three

Sinks and Voids

Sink marks are small depressions caused by the part shrinking while it is cooling. The usual cause of sinks is low injection pressure. If the pressure is too low, fewer molecules of plastic will enter the mold. When the part cools, the plastic molecules will shrink, leaving sink marks behind, as shown in Figure 6. Packing the part properly is the best defense against sink marks.

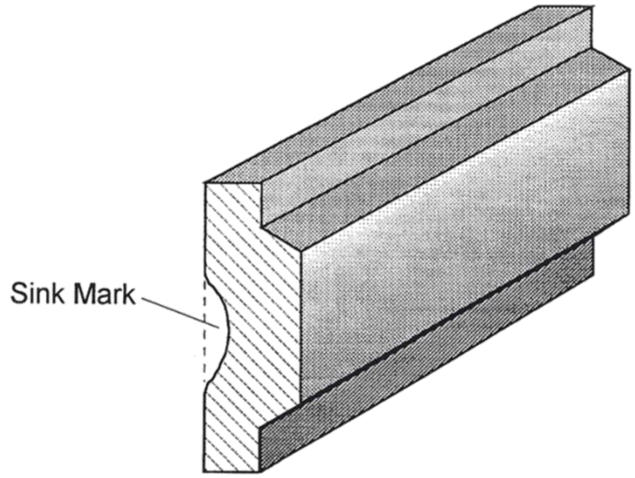

Figure 6 - Sink Mark

Sink marks can also form when the cooling time is too short, or the mold temperature is too warm. The part can only shrink as much as the cavity will allow. If the mold opens before the part has finished cooling, sections of the part can cool without the constraints of the cavity walls. Sink marks form because some sections of the part cool and shrink correctly, while other sections continue to shrink past their intended size. Likewise, if the mold is too hot when it opens, sections of the part will not be fully cooled. These sections will continue to cool and shrink after ejection, leaving sink marks behind.

It is also possible the melt temperature is too high. Hot plastic molecules take up more space than cool molecules. If the melt is too warm, fewer molecules of plastic enter and fill the cavity. When the part cools, the skin of the part solidifies first. As the rest of the part cools, it shrinks away from the already formed skin, leaving sink marks behind. The correct melt and barrel tempera-ture will keep the plastic warm enough to flow well into the cavity, but cool enough to avoid sink marks.

Exercise Three

Sink Marks

Find a part that has obvious sink marks. Record the following parameters.

Machine Number			
Mold Number			
Material			
Mold Temperature			
Transfer Pressure			
Hold Pressure			
Hold Time			

Instructor Date

Voids

Voids are bubbles of air or gas trapped in the plastic. Voids are usually caused by air, steam, or other gases being trapped in the melt. Voids are different from blisters. With blisters, the trapped gases make their way to the surface of the part. Voids are empty spots, or gases, trapped inside the part, like those illustrated in Figure 7.

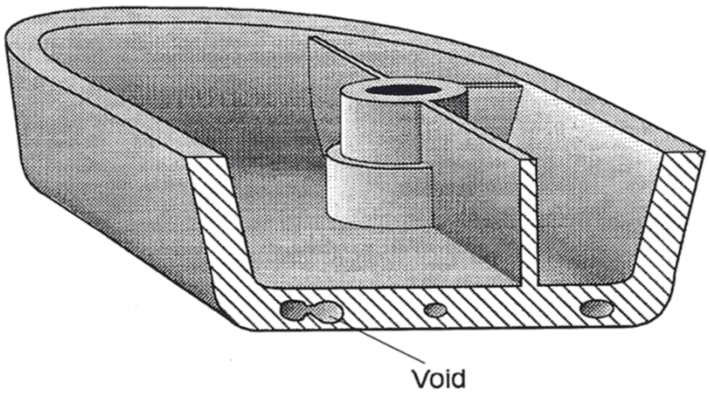

Figure 7 - Voids

Turbulence

When molten plastic becomes turbulent, it can trap tiny pockets of air in the part. Trapped air bubbles show up as voids. Fast injection speeds can cause turbulence. If the injection speed is too fast, the melt can tumble and crash as it enters the cavity. The melt front can impact against turns in the runners and gates, breaking the front. The unnecessary activity traps pockets of air in the part. Turbulent injection is illustrated in Figure 8.

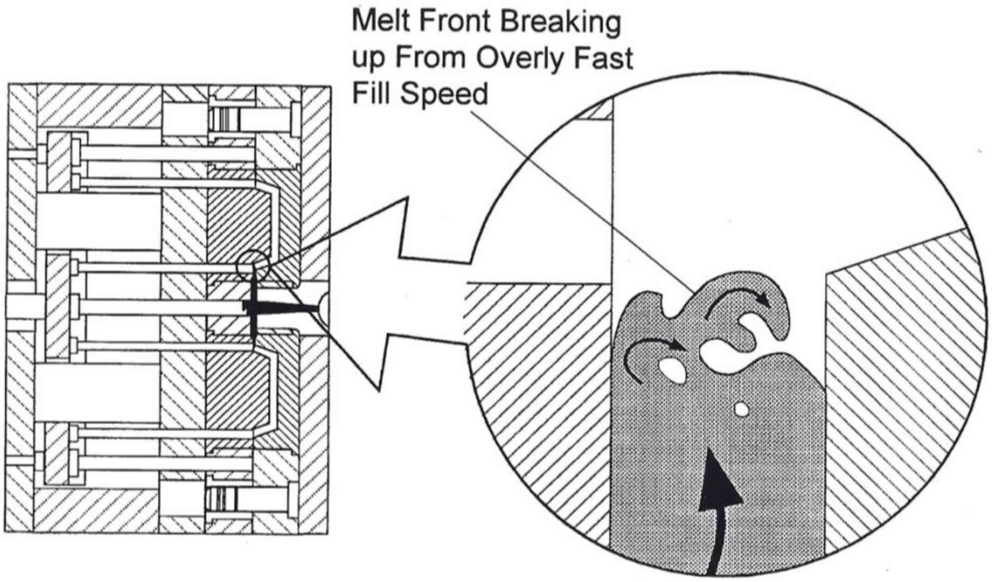

Figure 8 - Turbulent Flow

Even if the injection pressure is correct, high temperature or barrel temperature may cause the plastic to flow into the cavity too quickly. If the plastic is too runny, it can spray into the mold, trapping air.

Gates and Runners

Mold design and part design can easily cause voids. If the runners and gates are designed incorrectly, they may cause turbulence in the melt as it flows into the mold. Good designers use sophisti-cated flow analysis programs to predict how a melt front will navigate around comers, through gates, and into the cavity.

Poor Venting

Poor venting can also cause voids. Molds have vents to allow air in the cavity to escape. The air is replaced by the plastic that forms the part. If the air cannot escape properly, it will be trapped in the part, forming bubbles, or voids.

Low Back Pressure

During screw recovery, the plastic in the barrel is squeezed, but only a small amount. Plastic is mixed and melted in the barrel under slight pressure. Pressure in the barrel is maintained by the back pressure hydraulic valve. The slight pressure in the barrel helps squeeze out air bubbles before the next injection. If the back pressure is set too low, not all the air will be pressed out of the melt before injection.

Exercise Four

Air Bubbles

Locate a translucent or transparent part. Does the part have bubbles or voids in it? Where are the defects located?

Part Name	Location of Problem

Instructor   Date

Objective Four

Weld Lines

Weld lines, or knit lines, are lines in the part where two different fronts of molten plastic have come back together and welded. Weld lines usually occur around holes, or other flow obstacles.

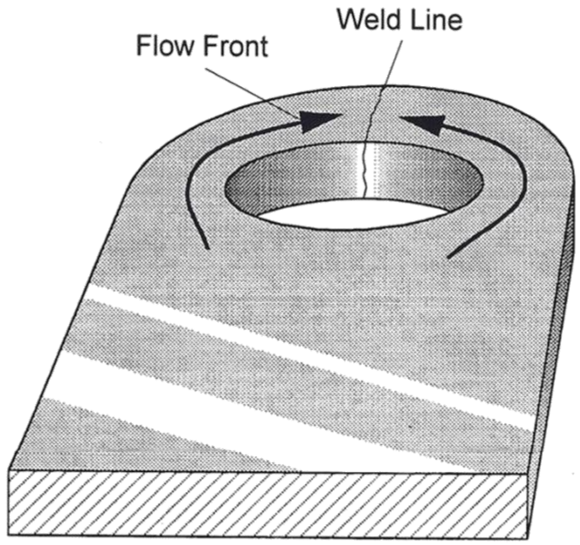

Figure 9 - Weld Line

If the melt temperature is too low, it may not stay molten long enough for both "waves" or fronts of plastic to surround the obstacle and rejoin. Figure 9 shows a knit, or "weld," line in a plastic part.

You can increase melt temperature by turning up the barrel heaters, increasing the barrel temperature (increasing the shear-ing effect), or even increasing the mold temperature. A warmer mold will allow the two fronts to join up while still molten.

Low injection pressures and speeds can also cause knit lines. Even if the melt and mold temperatures are correct, if it takes too long for the melt to flow around the hole or obstacle, the front edges of the melt will cool enough to prevent them from joining properly.

Exercise Five

Weld Lines

Try to find a part that has visible weld line on it. Record the following parameters off the molding machine.

Part Number			
Machine Number			
Highest Barrel Temperature			
Mold Temperature			
Injection Velocity			
Transfer Pressure			

Instructor Date

Objective Five

Jetting

Jetting is a crooked or snake like pattern on the surface of the molded part, like that shown in Figure 10. Jetting is usually caused by turbulence in the flow front of the resin.

Figure 10 - Jetting in a Molded Part

The most likely causes of jetting are fast injection speed, and excessive injection pressure. If the melt front is pushed through the gate too quickly, it can cause jet streams instead of the usual spreading out of the melt front. The individual streams cool differently from the rest of the melt, leaving jet shaped marks on the part. Bringing the injection speed down helps keep the melt front together.

Gates

Small or mislocated gates are a major cause of jetting. Gate design and location are usually an engineering and design func-tion. Most molding technicians do not have much control over gate size or placement. Some shops, however, design their gates into insert blocks that can be easily changed to adjust the size or length of the gates.

If the plastic is injected directly across a flat cavity surface, the surface drag can easily slow down part of the melt front. The plastic that slows down cools faster than the rest of the plastic. This leaves jet stream marks on the surface of the part.

Objective Six

Sticking Parts

Removing a stuck part from a mold is a tough job. Tools that make it easier to cut and pry a part from the cores can easily damage the mold face or cavity surfaces. There are many reasons why parts stick in the mold. Usually parts stick in the mold because of mold defects, or over-packing the cavity.

Problems with sticking parts are often related to over-packing. Over-packing means injecting too much melt into the cavity, as shown in Figure 11. Injecting too much melt might mean the injection pressure is too high, or the injection forward time is too long.

Mold Defects

If the mold is badly constructed, parts can stick. Some molds contain unintentional depressions or undercuts. Plastic fills around the depression and solidifies. Sometimes the undercut is strong enough, or far enough away from the ejector pins, to keep the part from ejecting.

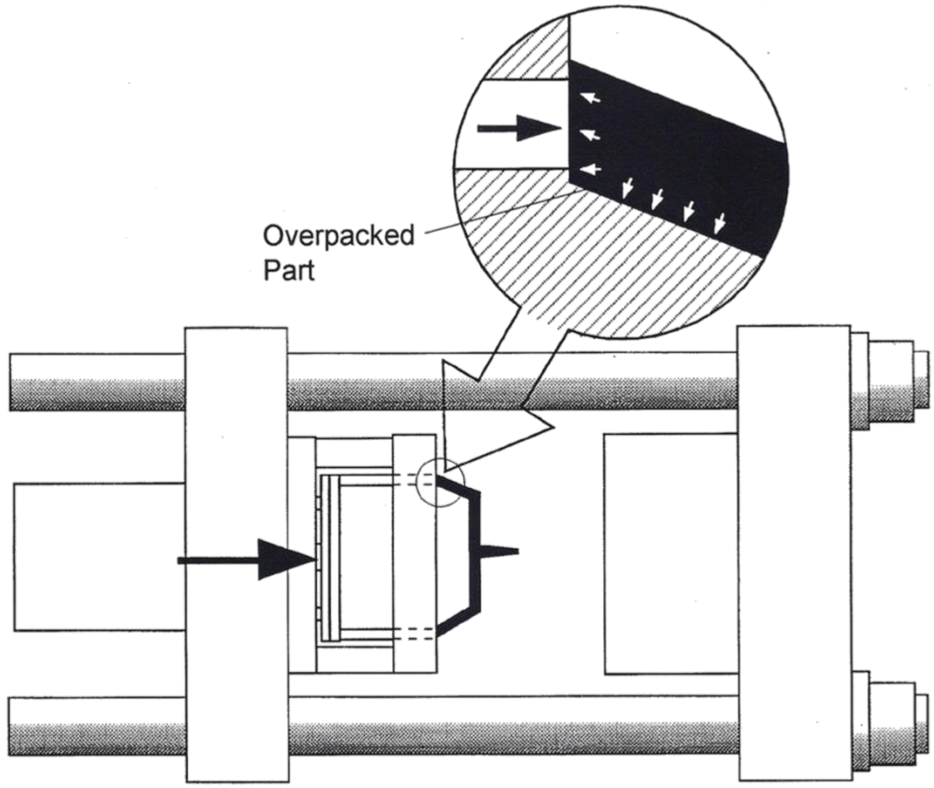

Figure 11 - Over-packed Part

Some parts are designed with a very slight draft. Draft is an angle or taper given to part surfaces to help them slide away from the walls of the core. If the draft angle is very slight, and the cavity is over-packed with material, there will be pressure on the side walls of the part. This pressure tends to lock the part into the mold.

More Cooling Time

If the part is not allowed to cool properly, the ejector pins can sometimes punch straight through. The part will then hang on the pins, and not fall out of the mold. Keep cooling time long enough to allow the part to solidify before the ejector pins try to push it off the core. If the part is allowed to cool too long it will shrink tighter on the cores. Mold temperature and cooling time need to be set so there is enough time for the part to cool and shrink properly before ejection, but not so the part shrinks too hard around the core.

Exercise Six

Sticking Parts

Locate and save the remains of several parts that were stuck in the mold. Record why the part was stuck in the mold and how long it took to remove it from the mold.

Part Number	Why was the part stuck?	How long to remove the part	Were pressures high or low?

Instructor Date

Sticking in the Sprue

Parts that stick in the sprue, or the cavity half of the mold, can be very difficult to remove. Usually there is no ejection system built into the stationary half of the mold. If a part sticks here, it can bring the machine down for hours.

Sticking in the sprue is usually caused by over-packing, or an oversized nozzle orifice. If the nozzle tip is misaligned or the wrong size, it may inhibit sprue removal. If the sprue becomes over-packed with material, pressure on the side walls will cause it to stay in the sprue bushing. You can also prevent over-packing the sprue by decreasing the holding pressure, or second stage time.

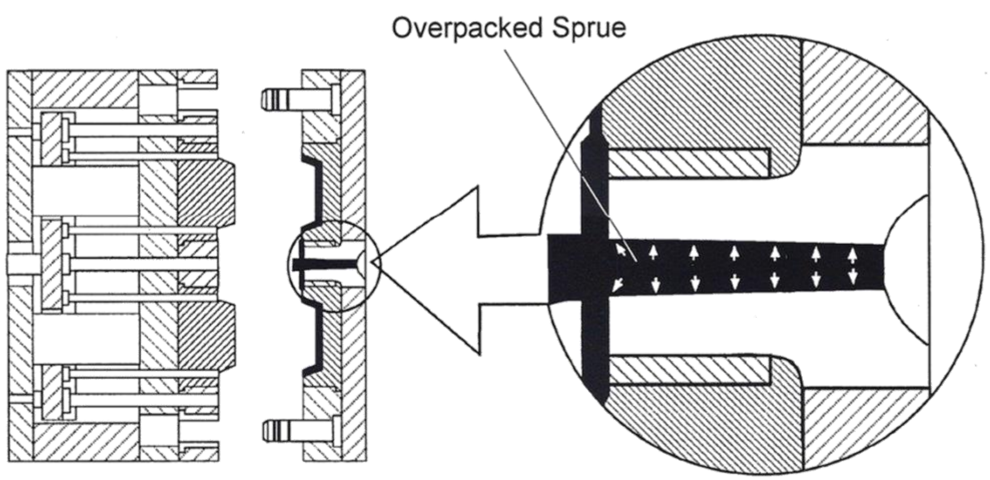

Figure 12 - Sprue Stuck in Sprue Bushing

Short Shots

During startup, some short shots may be so short that the ejector pins will not reach the part. In the early stages of anew production run, it is easy to make short shots where the screw only travels a slight distance. The part is left to cool in the mold without ejection.

Mold Sprays

A well-designed part in a good mold under the right process conditions should not need a mold release spray. Nevertheless, when parts begin to stick, most technicians immediately reach for a lubricant to help the parts release cleanly. Used sparingly, lubricants are a sensible response to a temporary problem. Examine parts that repeatedly stick in the mold for deeper design or process problems.

Self-Test

1. Most of the time, packing and filling defects come from:
   a. Problems with the mold design
   b. Problems with the process
   c. Problems with the quality of the melt
2. The most likely cause of flash is:
   a. Low back pressure
   b. Long cooling time
   c. High injection speed
   d. Low melt temperature
3. The most likely cause of short shots is:
   a. Large shot size
   b. Small shot size
   c. High injection pressure
   d. Low melt viscosity

4. Short shots are the opposite of:
   a. Voids
   b. Bubbles
   c. Distortions
   d. Long shots
   e. Flash
5. Short shots are more likely to occur when:
   a. The barrel temperature is too high
   b. The mold temperature is too high
   c. The shot size is too large
   d. The recovery time is too long
   e. None of the above
6. The usual cause of sink marks is:
   a. Low injection pressure
   b. High injection pressure
   c. Over-packing
   d. All of the above
   e. None of the above

7. Which of the following is NOT a likely cause of voids:
   a. Excess turbulence
   b. Low back pressure
   c. Excess venting
   d. Poor gate design
8. Parts usually stick in the mold because of mold defects or:
   a. Over-packing
   b. Under-packing
9. Fast injection speeds are a likely cause of:
   a. Jetting
   b. Turbulence
   c. Voids
   d. All of the above
10. A worn or misaligned check ring might cause:
   a. Flash
   b. Short shots

Glossary

Flash - extra plastic forced out of the mold, usually at the parting line.

Jetting - crooked or snake-like patterns on the part surface.

Land - the area on the surface of the mold about one inch around the outside of the part.

Melt front - the leading edge of the flow of plastic in the mold.
Short shot - a part that is not completely filled.

Sink marks - small depressions caused by shrinking while the part cools.

Voids - bubbles of air or other gases trapped in the part.

Weld lines - lines in the part where two different fronts of molten plastic have come together.