Common Injection Molding Defects, their Causes, and Repairs

When it comes to injection molding, small mistakes can cost companies large amounts of money. It is therefore imperative that any mistake, no matter how small, is caught early and repaired quickly. This takes technical expertise and high attention to detail. We will go over the most common mold defects, identify their respective causes, and cover their methods of repair. It is important to note that each mold defect has more than one possible cause and, alternately, multiple causes may bring about a single defect. For the purposes of this article, a mold *defect* refers to a damaged part produced by the injection molding process. An injection-molding *mistake* refers to the processing error that creates the mold defect. *Parts* always refer to injected plastics parts — not parts of injection tools or machinery. A greater understanding of injection molding errors and mistakes, as well as their respective causes and repairs, will also enable technicians and engineers to safeguard against such errors, and therefore prevent them from happening in the first place.

Here is a quick list of seven of the most common injection molding defects. Of course, this list is not exhaustive. The technician may encounter other defects than those presented and some of the defects encountered here may have alternate causes or methods of repair. It is our hope that this list proves a valuable primer as well as a handy reference for the aspiring technician to later expand upon as his or her knowledge develops.

1) Flow Lines
2) Sink Marks
3) Vacuum Voids
4) Short Shots
5) Burn Marks
6) Flash
7) Jetting

Flow Lines

Flow lines are essentially just as they sound: unintended visible lines 'flowing' on the surface of a finished plastic part. When viewed closely, flow lines appear as discolorations rather than indentations or abrasions. In most cases they are purely a cosmetic defect, but nonetheless constitute a 'bad part' (one that must be discarded). The underlying mistakes causing flow lines can furthermore cause other, more serious, defects.

Flow line discoloration is a result of inconsistent cooling times and temperatures. The molten plastic seen in flow lines have solidified at a faster or slower rate than the rest of the part, resulting in discoloration. Uneven cooling generally stems from two mistakes: unstable temperature and irregular speed variance. When parts are made too quickly, they do not have time to dry sufficiently, leaving streaks or flow lines. Parts do not dry properly at too low a temperature, regardless of speed variance.

In these cases, repairs closely resemble their causes. Either the injection speed or the temperature of molten plastic needs to be increased. The first step is to check these parameters, then adjust accordingly. If the temperature and speed are both optimal, the coolant may need to be adjusted, but this is seldom the case.

Sink Marks

Sink marks are small areas of parts that 'sink', forming depression in the plastic. Care needs to be taken in identifying such defects. They do not always appear immediately and thus can easily avoid detection. Imagine taking a cake out of the oven to cool and coming back a half hour later to find the center has sunken into itself. This is a useful analogy because it indicates the root cause that creates sink marks in plastic: uneven cooling. Sink marks usually occur where the plastic is at its thickest. The thickest section of the part needs the longest time to cool and therefore is the most prone to undercooling. Fortunately, molds are made with this in mind. An attempt at mass congruency within the mold cavity should already promote even cooling. This is not possible with all mold specifications, and therefore more care is needed when molding certain parts. In these cases, lowering the mold temperature can most often ensure even cooling time. If this does not work, the holding pressure may need to be increased, or, if all else fails, the mold itself might need to be customized. This customization is usually achieved by reducing the thickness of one of the wall sections.

Vacuum Voids

Vacuum voids are unique in that they only appear in translucent parts. Commonly referred to as air bubbles, these voids are pockets of air seen under the surface of clear plastic parts. Though they certainly classify as cosmetic damage, they can nonetheless compromise functional integrity. Familiarity with mistakes causing vacuum voids is integral for clear plastic parts and also their opaque counterparts. Although vacuum voids are only visible in clear plastic parts, they can exist in all parts, compromising opaque parts equally despite normal appearances.

Like sinks marks, vacuum voids are often the result of uneven cooling. What sink marks and vacuum voids have in common is the area of empty air underneath their surface. The difference is that the surface area covering vacuum voids is not sunken or depressed. The mistakes that cause vacuum voids are also different from those causing sink marks. Voids rarely result from the mistake of insufficient mold temperature; insufficient mold pressure is most often the instigator. Increasing holding pressure and/or holding time will usually rectify this situation. If this does not fix the problem, or the problem continues to resurface once fixed, a more permanent solution is often found by switching to a less viscous (more fluid) plastic.

Short Shot

A short shot occurs when not enough molten plastic is injected, or shot, into the mold. In other words, it falls short. This may or may not result in an underfilled part (a part that has obviously missing pieces as a result of insufficient plastic injection). Occasionally the part is able to hide its missing plastic in the form of air bubbles, sink marks, or other less severe problems. In any case, short shots always produce bad parts due to their incompleteness. Many different mistakes produce short shots, so they are notoriously difficult to troubleshoot. There are, however, a few common causes, the most common being incorrect calibration. If re-calibrating the machine does not fix the problem, the issue could be the material. A less viscous plastic may again provide the solution by achieving more forgiving fluidity. Even if the current plastic is not the ultimate problem, a more fluid plastic may save you the headache of constant re-calibrations. This is an especially viable solution when more complicated repairs must be made, for example adjustments to gas ventilation or material feed.

Burn Marks

For previous generations, burns marks — black spots on the exterior of finished parts — was a ubiquitous problem with a myriad of causes. Nowadays, the defect generally boils down to one mistake: pre-existing burn marks in the plastic granulate. If burn marks show up in the finished product, your first step should be to check the plastic granulate. Should the plastic granulate pass inspection, the problem could be the overheating of trapped air. In this case, you can obviously reduce the mold temperature or, if necessary, adjust the gas venting to fix the problem. You most likely will never face this issue, because such mistakes generally produce other defects far earlier than they produce burn marks — again establishing plastic granulate as the primary culprit for the burn mark defects.

Flash

Flash can be thought of as the opposite of a short shot. Its correlate is an over-filled shot: when too much, rather than too little, plastic is released into the shot cavity. This extra plastic is usually revealed in various undesired appendages to the finished part, such as wispy strings or bulbous protrusions. If, indeed, too much plastic is released into the part, this is usually a byproduct of excessive pressure. It is not always excessive molten plastic, but sometimes insufficient clamping force, that allows some of the molten plastic to stray from the part during the molding process ('over-filled', in this case, is technically a misnomer). Generally, both problems are easy to spot and simple to repair. In the former case, the likely scenario is that the mold has reached or is reaching the end of its life cycle. In the latter case, the pressure simply needs to be adjusted downwards.

Jetting

Jetting derives its namesake from the root cause responsible for its physical defect: high velocity injection speed. The plastic jets forwards so fast that a wave-like imprint is solidified in the final product much like water that is frozen in motion. The cause is almost exclusively the high viscosity of the molten plastic; however, this usually combines with an insufficient melt temperature to create the jetting effect. The melt temperature should initially be increased and if jetting persists, the injection speed can be indirectly lowered through increasing the gate size. In the rare case that neither solution works, the gate design must be optimized.

These are some of the most basic mold defects, their causes and solutions. You should now be in a good position to handle most defects that come your way, even if we haven't covered them all. Other examples include warpage, scratches, and delamination. Luckily these are not as common as they once were, which is why they have been omitted. Some defects, while not industrially common, may prevail in your plant nonetheless. It is our hope that this guide has prepared you for the basics. The more senior technicians at your facility can educate you in company specifics.