

PVC Injection Molding

Top 5 Troubleshooting Guidelines

Troubleshooting for injection molding requires a systematic approach with proper observation and documentation.

Best practices include:

- Define the problem
- Develop a method to isolate the problem
- Test one item at a time to verify results.
- Monitor the final solution to verify the problem has been solved.
- Document the solution - this can ease similar problems in the future.

While each molded part is unique and there is no universal remedy for all molding defects, there are common quality-related issues which can be resolved with simple adjustments to either the machine conditions or the mold design. Following are five (5) common injection molding defects and their probable causes. This resource can assist you in determining and correcting the issue to ensure a quality molded part.

While this guide can be helpful once a defect has occurred, many common molding issues can be avoided by a proper setup procedure including reviewing historical information and verifying all job settings in advance. If you have specific questions about processing your molded part, please feel free to contact us.

Defect	Caused by Mold	Caused by Machine
Sink Marks Small depressions in the material resembling dimples	-Inadequate cooling time -Mold temp too high -Gates or runners too small	-Mold temp too high -Gates or runners too small -Improper gate location
Flow Lines Discolored lines or patterns on the finished product	-Inadequate injection pressure -inadequate residence time -Barrel temp too low	-Mold temp too low -Gates or runners too small -Inadequate venting
Splay Also called “silver streaks”. A splash-like appearance or spray pattern on the surface of the molded part.	-Barrel temp too high -Excessive screw speed -Nozzle too small, too hot, or obstructed	-Obstruction in the gate or runner -Gate too small -Cracks in mold
Burn Marks Small dark brown or black discolorations on the surface of the molded part.	-Injection speed or pressure too high -Barrel temp too high -Screw speed too fast	-Gates too small -Inadequate venting -Clamping force too high
Flash The thin layer of plastic that flows outside of the cavity where the two halves of the injection mold meet.	-Clamping force too low -Injection pressure too high -Injection speed too high	-Clamping force too low -Damaged mold -Inadequate mold supports

This document is intended to serve as a reference guide only. Alphagary/Mexichem makes no warranty, express or implied, that all materials under all conditions will behave as represented here.