Philosophy of Troubleshooting BMC Injection Molding Problems

There is no "cure-all" for eliminating molding problems. Every mold is distinctive and has its own peculiarities, even molds that are exact duplicates. The same holds true for molding presses. Therefore, what works for one mold or press may not work for another.

Problems can differ with conditions such as the shop climate, material or equipment resulting in numerous variables, many of which are not obvious and are difficult to identify. The troubleshooting suggestions that follow have been used to resolve the stated problem. However, due to the reasons mentioned above, several of them may seem to be contradictory.

The most important thing to remember is make a reasonable adjustment and carefully observe the results. Document the change and keep track of the response by comparing parts from before and after the adjustment, to determine if the effect is favorable. This will lead to other changes that should eventually produce a solution to the problem.

Thermoset materials cure because of a chemical reaction. Temperature has a major role in the speed of the reaction and to a lesser degree so do pressure and time. Therefore, all three items need to be verified before troubleshooting any molding problem.

For **injection molding**, this means there are several items on the mold and press that should be checked before troubleshooting any molding problem. If problems are found, they should be corrected, and the parts re-evaluated before proceeding.

Mold - Check the actual steel temperature of the mold and its uniformity across the entire surface of both mold halves. Be sure to use a Calibrated Pyrometer and Surface Probe. Compare the readings with the temperature readings of the last time the mold was processed. The data should be similar or close to it. If not, investigate why there are differences. Correct as necessary.

Press - Compare process parameters (primary & holding pressures, injection time, hold time, back pressure etc.) against process parameters from previous run. The data should be similar or close to it. If not, investigate why there are differences. Correct as necessary.

Injection Processing Problems

Please **NOTE** the following:

- Various processing changes are suggested as possible solutions for the different problems encountered during the molding of thermoset parts. In general, these changes should not exceed the recommended ranges presented in the previous sections on injection startup procedures.

- This troubleshooting guide lists the processing problems in alphabetical order.

- For each problem, the possible solutions are listed in the order of the most frequent to the least frequent solution for that problem.

- Change only one processing variable at a time. Allow the press to cycle at a minimum of 3 cycles and then thoroughly evaluate results before changing another processing variable.

- In order to resolve any given processing problem, a combination of the possible solutions may need to be applied.

- PLENCO's Technical Service Group is available to assist you in troubleshooting processing problems.

CRAZE CRACKING - Hairline cracks that appear on the surface of the part but do not propagate through the entire part thickness.

POSSIBLE SOLUTIONS

1. Check to be sure parts are well packed, with vent tailings with a minimum of a 0.125" cushion.
2. Decrease ejection speed.
3. Decrease ejection pressure.
4. Check for undercuts and reverse draft. Address as necessary.
5. Increase clamp tonnage.
6. Increase holding pressure.
7. Increase holding time.

CONTAMINATION - Any visible foreign material on the surface of the molded part.

POSSIBLE SOLUTIONS

1. Check all unmolded material for foreign matter.
2. Check all equipment used in molding the material for potential sources of contamination. Clean as needed.
3. Check for air borne particulates from other processes and eliminate their source. Segregate area if possible.
4. Contact material supplier.

DIESELING - A burnt spot on part, often accompanied by non-fills.

POSSIBLE SOLUTIONS

1. If mold is vacuum vented, check system to insure that it is pulling a minimum of 21" of Hg in the mold. If it is not, resolve problem with vacuum system.
2. Decrease injection speed.
3. Decrease primary pressure.
4. Adjust primary to holding pressure transfer position to start sooner in the injection stroke.
5. Decrease mold temperature.
5. Modify ejector pin ends to include a stove pipe design to allow material to flow down the ejector pin end.
6. If mold is not vacuum vented, if possible, install vacuum venting.

DRAG MARKS - Surface scratches that occur during part ejection.

POSSIBLE SOLUTIONS

1. Check the area(s) on the mold where there drag marks are occurring for burrs, chips or undercuts. Polish area(s) to remove them.
2. Check guided bushings for wear.

DULL APPEARANCE - The surface of the part has a hazy or satiny appearance instead of a glossy appearance.

NOTE: Make sure part is completely filled out and mold is not stained.

POSSIBLE SOLUTIONS

1. Increase mold temperature.
2. Increase barrel temperature.
3. Increase cure time.
4. Check to be sure parts are well packed, with vent tailings and a cushion.
5. Polish dull area(s) of the mold.
6. Check condition of mold plating. Polish and re-plate the mold.
7. Contact material supplier.

FLASH (EXCESSIVE) - Parts where the flash is thicker than 0.15 mm (0.006") or with flash extending out into the land areas are considered to have excessive flash.

POSSIBLE SOLUTIONS

1. Decrease shot size.
2. Decrease primary pressure.
3. Decrease holding pressure.
4. Decrease injection speed.
5. Increase clamp tonnage.
6. If using a dual screw BMC stuffer, decrease the RPM's & torque % of both screws.

FLOW LINES - Visible lines on the surface of the part that show the flow pattern of the material as it filled the cavity.

POSSIBLE SOLUTIONS

1. Decrease injection speed.
2. Decrease holding pressure.
3. Check to be sure parts are well packed, with vent tailings and a cushion.
4. Add a flash well at the end of the vent(s) to allow material to flow into the flash well and better knit together in the cavity.
5. If possible, relocate the gate perpendicular to the part instead of parallel flow to its geometry.

GAS BURNS - A porous, dull, discolored and sometimes scorched area on the surface of a part.

POSSIBLE SOLUTIONS

1. If mold is vacuum vented, check if system is pulling a minimum of 21" Hg in the mold. If not, resolve problem with vacuum system.
2. Check the vents and correct as needed.
3. Decrease stock temperature by decreasing backpressure and/or barrel temperature.
4. Decrease injection speed.
5. Decrease injection and holding pressures.
6. Decrease mold temperature.
7. Decrease clamp tonnage.
8. Increase gate & runner size.

INJECTION TOO SLOW - During the injection portion of the molding cycle, the material does not fill the mold in the recommended injection time.

POSSIBLE SOLUTIONS

1. Increase primary pressure.
2. Increase injection speed.
3. Increase nozzle temperature.
4. Increase front barrel zone temperature.
5. Increase the size of the gate – a higher fiberglass level within certain BMC formula's, may affect a materials ability to flow through a smaller size gate.

KNIT LINES – Areas where multiple flow fronts meet but do not fuse or knit together.

POSSIBLE SOLUTIONS

1. Increase the size of the gate and if possible, relocate it.
2. Decrease primary pressure.
3. Decrease injection speed.
3. Decrease mold temperature.
4. Add a flash well at the end of the vent(s) to allow material to flow into the well and better knit together in the cavity.

LAKING – Irregular dull areas on the surface of the part.

POSSIBLE SOLUTIONS

2. Increase clamp tonnage.
3. Check to be sure parts are well packed, with vent tailings with a minimum of a 0.125" cushion.
4. Adjust primary to holding pressure transfer position to start sooner in the injection stroke.
5. Increase holding pressure.
6. Increase hold time.
6. Check parting line for wear or damage and repair as needed.

NONFILLS OR SHORT SHOTS – Areas of surface porosity due to parts not being completely filled out.

POSSIBLE SOLUTIONS

1. Increase shot size.
2. Increase clamp tonnage.
3. If mold is vacuum vented, check system to insure that it is pulling a minimum of 21" of Hg in the mold. If it is not, resolve problem with vacuum system.
4. Adjust primary to holding pressure transfer position to start later in the injection stroke.
5. Increase the size of the gate – a higher fiberglass level within certain BMC formula's, may affect a materials ability to flow through a smaller size gate.

PRE CURE - Localized areas of dull rough porosity.

POSSIBLE SOLUTIONS

1. Decrease mold temperature.
2. Decrease barrel temperature.
3. Increase injection speed.
4. Increase primary pressure.
5. Decrease injection delay time.

SCREW PICKUP IS ERRATIC - During the screw return portion of the molding cycle, the screw does not return to its set point at a uniform rate.

POSSIBLE SOLUTIONS

1. If using a dual screw BMC stuffer, increase the RPM's & torque % of both screws.
2. Decrease injection screw speed.
3. Increase rear zone barrel temperature.
4. Increase back pressure.
5. Check the screw and barrel for wear and if necessary, recondition or replace them.

SCREW PICKUP IS TOO SLOW - During the screw return portion of the molding cycle, the screw takes too long to return to its fully retracted position at a uniform rate.

POSSIBLE SOLUTIONS

1. If using a dual screw BMC stuffer, increase the RPM's & torque % of both screws.
2. Increase injection screw speed.
3. Decrease back pressure.
4. Increase rear zone barrel temperature.

SHRINKAGE – There are two problems that will cause a part to not meet its dimensional requirements, the part has excessive shrinkage (undersize) or the part has insufficient shrinkage (oversize).

POSSIBLE SOLUTIONS FOR EXCESSIVE SHRINKAGE (UNDERSIZE)

1. Increase primary pressure.
2. Increase holding pressure.
3. Increase mold temperature.
4. Increase clamp tonnage.
5. Increase cure time.
6. Contact material supplier.

POSSIBLE SOLUTIONS FOR INSUFFICIENT SHRINKAGE (OVERSIZE)

1. Decrease mold temperature.
2. Decrease primary pressure.
3. Decrease holding pressure.
4. Decrease clamp tonnage.
5. Decrease cure time.
6. Contact material supplier.

SINK MARKS – Slight depressions on the surface of the part that resemble dimples.

POSSIBLE SOLUTIONS

1. Increase shot size.
2. Increase mold temperature.
3. Increase primary pressure.
4. Increase holding pressure.
5. Decrease injection speed.
6. Adjust primary to holding pressure transfer position to start sooner in the injection stroke.
7. Add a flash well at the end of the vent(s) to allow material to flow into the well and better knit together in the cavity

SPRUE STICKING - When the mold opens during the molding cycle, the sprue will not release from the sprue bushing and will remain on the fixed half of the mold.

POSSIBLE SOLUTIONS

1. Make sure the orifice of the sprue bushing is larger than the nozzle orifice.
2. Check sprue bushing for damage or wear. Replace it.
3. Decrease primary pressure.
4. Decrease holding pressure.
5. Decrease clamp slow open speed.
6. Check sprue puller design and revise as needed.

STICKING IN MOLD - Runner, and/or part will not release from the mold and apiece or all of the part will remain stuck until it is manually removed.

POSSIBLE SOLUTIONS

1. Add under cuts in the moveable mold half.
2. Decrease clamp slow open speed.
3. Decrease holding pressure.
4. Increase mold temperature.
5. Decrease shot size.
6. Increase cure time.
7. Check sprue puller design and revise as needed.
8. Check condition of mold plating and re-plate if necessary. If mold is unplated, polish or plate it.

WARPAGE - Part is twisted or warped rather than straight or flat. This can occur when **ejected from the mold** or **after cooling**.

POSSIBLE SOLUTIONS FOR PART WARPAGE WHEN EJECTED FROM MOLD

1. Add undercuts to hold parts in the movable half of the mold until they are ready to be ejected.
2. Check dropping of parts from the mold or observe the part picker to see if the parts are being deformed.
3. Increase mold temperature.
4. Increase cure time.
5. Contact material supplier.

POSSIBLE SOLUTIONS FOR PART WARPAGE AFTER COOLING

1. Increase mold temperature.
2. Increase barrel temperature.
3. Increase back pressure.
4. Decrease injection speed.
5. Increase cure time.
6. Use cooling fixture(s) to hold the parts flat as they cool.
7. Increase the size of the gate and if possible, relocate it.
8. Contact material supplier.

WOOD SCREWING - During the screw return portion of the molding cycle, the screw will not pickup enough material as it rotates back to its set point.

POSSIBLE SOLUTIONS

1. Check stuffer RPM and Torque set points. Increase set points to ensure material feed is keeping up with injection screw RPM.
2. Piston stuffer, increase pressure if possible.
3. Increase rear barrel zone.
4. Decrease screw speed.
4. Decrease back pressure.
5. Inspect BMC screw and check ring to ensure proper function.
6. Contact material supplier.