



Philosophy of Troubleshooting Injection Molding Problems

There is no "cure-all" for eliminating molding problems. Every mold is distinctive and has its own peculiarities, even molds which 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 successfully 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 as a result 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 mold temperature and its uniformity across the entire mold face and compare with what the temperature was the last time this mold was run in this press. They should be the same or very close to it. If not, try to determine why there are differences before starting to mold.

Press - Check actual settings for injection pressure, back pressure, holding pressure, injection speed, injection time, screw speed and return time with the settings used the last time this mold was run in this press. They should be the same or very close to it. If not, try to determine why there are differences before starting to mold.

Injection Processing Problems

Please **NOTE** the following:

- Increased cycle time should not be used as a solution for a molding problem, except as the very last resort to maintain the integrity of the molded parts.
- 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 cycles and then thoroughly evaluate results before changing another processing variable.
- Please keep in mind, that 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.

BALL & SOCKET - An internal circular failure pattern observed on injection molded parts after breaking of the thickest cross-sectional area. One surface will resemble a ball and the other surface will resemble a socket. The texture and color of the socket area will differ from that of the ball.

POSSIBLE SOLUTIONS

1. Decrease injection speed.
2. Increase injection and holding pressures.
3. Decrease mold temperature.
4. Increase the size of the gate and if possible relocate it.

BULGE OPPOSITE INSERT - Bulges that appear on the side opposite and directly over molded-in inserts.

POSSIBLE SOLUTIONS

1. Heat inserts to mold temperature before using.
2. Use a reduced size insert.
3. Increase mold temperature.
4. Increase stock temperature by increasing back pressure and/or barrel temperature.

CURE BLISTER - Area of gas entrapment (blister) caused by not completely curing the part before removing it from the mold. This generally shows up as a bulge on opposite sides of the thickest cross-sectional area of the part. When broken open, there will be a large void in the center of the bulge.

POSSIBLE SOLUTIONS

1. Increase stock temperature by increasing back pressure and/or barrel temperature.
2. Increase mold temperature.
3. Increase cure time.
4. Decrease injection speed.

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 stock temperature by increasing back pressure and/or barrel temperature.
3. Check condition of mold plating and re-plate if necessary. Polish and re-plate the mold.
4. Polish the mold.

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 injection and holding pressures.
3. Increase stock temperature by increasing back pressure and/or barrel temperature.
4. Increase mold temperature.
5. Check parting line for wear or damage and correct as needed.
6. Increase clamp tonnage if possible.

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. Increase injection speed.
2. Increase injection and holding pressure.
3. Increase holding time.
4. Decrease mold temperature.
5. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
6. Increase the size of the gate and if possible relocate it.

HARD SPOTS (PRECURE) - Slight bumps on the surface of the part that are usually uneven, pointed, and rough and have definite outlines.

POSSIBLE SOLUTIONS

1. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
2. Decrease injection and holding pressures.
3. Decrease injection speed.
4. Decrease mold temperature.
5. Check sprue tip for "soft bulb" and adjust conditions as needed.

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 injection pressure.
2. Check injection throttle position and if possible, increase its setting.
3. Increase stock temperature by increasing back pressure and/or barrel temperature.
4. Increase or decrease mold temperature.

MOLD STAINS - A build up of volatiles on the molding surface that will cause that surface area of the part to be dull and pit marked. This can eventually lead to part discoloration and parts sticking in the mold.

POSSIBLE SOLUTIONS

1. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
2. Polish the mold.
3. Increase stock temperature by increasing back pressure and/or barrel temperature.
4. Increase mold temperature.
5. Decrease injection and holding pressures.
6. Decrease injection speed.
7. Decrease clamp tonnage.

MOTTLED SURFACE APPEARANCE - A non-uniform coloring or texture on the surface of the part.

POSSIBLE SOLUTIONS

1. Increase shot size.
2. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
3. Decrease mold temperature.

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

POSSIBLE SOLUTIONS

1. Increase shot size.
2. Increase stock temperature by increasing back pressure and/or barrel temperature.
3. Increase injection and holding pressures.
4. Decrease mold temperature.
5. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
6. Check the screw and barrel for wear and if necessary recondition or replace them.

NOZZLE FREEZE UP - Curing of the compound in the nozzle during the molding cycle which prevents the screw from injecting the material on the next cycle.

POSSIBLE SOLUTIONS

1. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
2. Decrease mold temperature.
3. Check sprue tip for "soft bulb" and adjust conditions as needed.

ORANGE PEEL - Surface appearance that looks like an undersurface craze or numerous small ripples and resembles the skin of an orange.

POSSIBLE SOLUTIONS

1. Increase injection and holding pressures.
2. Increase stock temperature by increasing back pressure and/or barrel temperature.
3. Increase mold temperature.
4. Decrease injection speed.

RUBBERY PARTS OR RUNNER - Parts and/or runner that are rubbery on ejection from the mold.

POSSIBLE SOLUTIONS

1. Increase mold temperature.
2. Increase stock temperature by increasing back pressure and/or barrel temperature.
3. Request from material supplier a version of the material that has a lower deflection specification

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

POSSIBLE SOLUTIONS

1. Increase stock temperature by increasing back pressure and/or barrel temperature.
2. Decrease screw speed.
3. Check material feed from hopper.
4. 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. Increase screw speed.
2. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
3. Increase barrel temperature of the feed zone.

SHRINKAGE - There are two problems which 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 injection and holding pressures.
2. Increase mold temperature.
3. Increase stock temperature by increasing back pressure and/or barrel temperature.
4. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
5. Increase cure time.

POSSIBLE SOLUTIONS FOR INSUFFICIENT SHRINKAGE (OVERSIZE)

1. Decrease mold temperature.
2. Decrease holding pressure.
3. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
4. Decrease cure time.

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 injection and holding pressures.
4. Decrease injection speed.
5. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")

SKIN BLISTERS - Small areas of gas entrapment (blisters) on the surface of the part that when broken open, appear to have occurred just under the "skin" of the part. They are generally spaced randomly about the surface of the part and many times will appear on only one surface.

POSSIBLE SOLUTIONS

1. Decrease injection speed.
2. Increase shot size.
3. Decrease mold temperature.
4. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")

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

POSSIBLE SOLUTIONS

1. Make sure the sprue bushing orifice is larger than the nozzle orifice. Also check sprue bushing for damage or wear and either repair or replace it.
2. Decrease injection pressure.
3. Decrease holding pressure.
4. Check sprue tip for "soft bulb" and adjust conditions as needed.
5. Check sprue puller design and revise as needed. (See Section #6, "Thermoset Injection Mold Design Tips")

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

Please Note: When molding single stage phenolic molding compound or granular thermoset polyester compounds, Plenco recommends that the mold be chrome plated, since these materials have a tendency to stick to non-chromed surfaces.

POSSIBLE SOLUTIONS

1. Check mold for wear and correct as needed.
2. Decrease holding pressure.
3. Increase mold temperature.
4. Decrease shot size.
5. Check condition of mold plating and re-plate if necessary.
6. Increase cure time.

SUBGATES STICKING IN MOLD - Subgates will not release from the mold and a piece or all of it will remain stuck until it is manually removed.

POSSIBLE SOLUTIONS

1. If a new mold, check subgate design and correct as needed.
2. If an existing mold, check subgates for damage or wear and repair or replace as needed.
3. Request from material supplier a version of the material that has a higher deflection specification.

TRAPPED GAS (BURN MARK) - 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 mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
3. Decrease stock temperature by decreasing back pressure and/or barrel temperature.
4. Decrease injection speed.
5. Decrease injection and holding pressures.
6. Reduce mold temperature.
7. Decrease clamp tonnage.
8. Increase gate & runner size.

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. Check mold for wear and correct as needed.
2. Check condition of the mold plating and re-plate if necessary. If mold is un-plated, polish or plate it.
3. Add undercuts to hold part until properly ejected.
4. Check drop of parts from mold or observe the part picker to see if the parts are being deformed.

POSSIBLE SOLUTIONS FOR PART WARPAGE AFTER COOLING

1. Increase mold temperature.
2. Increase stock temperature by increasing back pressure and/or barrel temperature.
3. Decrease injection speed.
4. Check mold venting and correct as needed. (See Section #6, "Thermoset Injection Mold Design Tips")
5. Increase cure time.
6. Use shrink fixture to hold the parts flat as they cool.
7. Increase the size of the gate and if possible relocate it.

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

POSSIBLE SOLUTIONS

1. Increase back pressure.
2. Increase rear zone barrel temperature.
3. Decrease screw speed.

Date Printed: November 10, 2015

Date Revised: December 3, 2014

Supersedes Revision Dated: August 15, 2007

This information is suggested as a guide to those interested in processing Plenco Thermoset molding materials. The information presented is for your evaluation and may or may not be compatible for all mold designs, runner systems, press configurations, and material rheology. Please feel free to call Plenco with any questions about PLENCO molding materials or processing and a Technical Service Representative will assist you