

TROUBLESHOOTING GUIDE FOR INJECTION MOLDING



Short shots:

- Increase the size of the shot or of the material base and assure its consistency. Do not use reverse pressure since it may cause inconsistency in the plasticization of the shot.
- Increase the injection and/or the compacting pressure.
- Increase the injection speed.
- Increase the time of the front screw and/or adjust the transference point of the filling stage to the compacting stage. Adjust it in such way that 95% of the part may come out at the filling stage.
- Gradually increase by 10°C the cast temperature.
- Check to see if the nozzle or cavity entries are clogged.
- Check the anti-return valve looking for leakages.
- Increase the temperature of the mold.
- Clean or improve the venting of the mold.
- Enlarge the entry to the mold and/or the diameter of the feeding ducts and decrease the length of the entry.
- Use a higher index flow resin.
- Use a machine with greater injection capacity.
- If it is a new design of a part: increase the thickness of the wall, add flow leads in the piece, relocate the injection points or increase their number. Use injection simulation programs for improving the design.

Contraction marks shrink holes and vacuums:

- Increase the compacting or sustained pressure.
- Increase the time of the sustained pressure or the time of the front screw. Verify that this time coincides with the sealing of the cavity entry.
- Decrease the mold temperature. Exception: with thick wall parts increase the mold temperature and increase the cooling time.
- Decrease the casting temperature.
- Adjust the injection speed (it should be typically reduced).
- Verify that the cooling ducts do not have deposits or embeddings.
- Check the nozzle heating system or if same is obstructed. If possible, enlarge the nozzle diameter.
- Adjust the dose and the material mattress. Verify that the material mattress is consistent,

in order to accomplish this, make sure there is reverse pressure and that the anti-return valve is working properly.

- Experiment with additional post-molding cooling for thick pieces.
- Enlarge the cavity entry and/or the feeding ducts and decrease the length of the entry.
- Increase the mold entry and the feeding ducts size and the nozzle.
- Increase the entry diameter and decrease the long of it.
- Change the entry position of the mold.
- In the part design: locate the cavity entries in the thicker parts, include insertions or inner bulges between 50 and 60% of the wall thickness for those visible wall surfaces.

Irregularities or rough edges:

- Verify that the established closing force is reached before starting the injection.
- Increase the closing force.
- Decrease the dosage (the size of the shot) or the material mattress.
- Reduce the injection pressure. Take the pressure to zero in the sustained pressure stage (compacting) in order to verify if the forming of irregularities or rough edges occur during the filling stage.
- Reduce the casting temperature.
- Use a very low sustained pressure at the beginning of the compacting stage, increase it afterwards.
- Reduce the injection speed before reaching the maximum pressure in that cavity.
- Move forward the second stage transfer (compacting) or change the piston movement control method (controlled by the pressure speed).
- Check the alignment and parallelism of the mold and rectify the surfaces or support pillars if necessary.
- Check the cavity sealing area in the mold. Check the areas of the partition line, insertions, etc., verifying there are appropriate settlements or fittings.
- Decrease the thickness of the ventilation ducts and increase the amount or the length of same.
- Check if the correct resin is being used.

TROUBLESHOOTING GUIDE FOR INJECTION MOLDING



Superficial spots adjacent to the injection points:

- Reduce the injection speed.
- Reprogram the injection speed profile (if the injector has this capacity), by reducing the casting speed when passing the injection point and, then, accelerating it in order to complete the filling of the cavity and thus avoiding short shots.
- Increase the casting temperature.
- Increase the mold temperature.
- Increase the nozzle temperature.
- Increase the cavity entrance.
- Verify that the mold has a cold well.

Burns and discoloration:

- Reduce the injection speed.
- Reduce the residence time in the barrel, decrease the injection cycle.
- Decrease the casting temperature.
- Decrease the reverse pressure.
- Reduce the rotation speed of the screw.
- Slowly decrease the closing pressure.
- Eliminate the possible contamination with other resins or additives.
- Clean the mold ventilation. Modify the mold if it does not have ventilation.
- Clean the ejector pins.
- Check whether there is accumulates resin in the no-return valve.
- Verify that the thermocouple and/or the heating bands are working properly.
- Change the number and/or the location of the injection points. Use injection simulation programs to locate the best placement of the injection points and thus reducing or eliminating the trapped air which produces the diesel (burns) effect.

Orange peel skin or cellulite:

- Increase the injection pressure.
- Decrease the injection speed.
- Increase the casting temperature.
- Increase the mold temperature.
- Verify if the resin used is contaminated.

Non-homogeneous color or with streaks:

- Increase the reverse pressure.
- Reduce the rotation speed of the screw.
- Reduce the cooling of the hopper.

- Inspect the color concentrate in order to verify if it is homogenous.
- Use a mixing screw or mixing nozzle.
- Increase the mattress in the material dose.
- Verify if the resin used is contaminated.

Superficial defects:

- Periodically clean the mold.
- Improve the ventilation, superficial finish or the polish of the mold.
- Decrease the residue deposit in the mold. In order to do this, decrease the casting temperature, reduce the residence time in the barrel, decrease the injection speed.
- Check if there are any hot spots in the mold.
- Assess the additives package of the resin.

Ejector pin marks:

- Decrease the ejection force of the piece.
- Increase the cooling time or decrease the temperature of the mold.
- Level route of the ejector pins.
- Increase the size or amount of the ejector pins.
- Relocate the ejector pins towards places with higher resistance.

Flow marks (jetting).

- Reduce the injection speed. If possible, program the profile of the velocities so that the flow becomes slow when the entry area is forming and afterwards increase the speed.
- Increase the casting temperature.
- Increase the mold temperature.
- Increase the depth of the cavity entry. A depth between 60 and 80% is recommended.
- Relocate the injection points in the mold so that the flow is facing the wall of the mold (the cast material does not have a large space in front of it).
- Change to a wider, fan type cavity entry.

Welding line excessively marked or too weak:

- Increase the injection and/or compacting pressure.
- Increase the time of the sustained (compacting) pressure.
- Increase the injection speed.
- Increase the casting temperature.
- Increase the temperature of the mold.
- Improve the ventilation of the mold.

TROUBLESHOOTING GUIDE FOR INJECTION MOLDING



- Adjust the transfer point of the filling stage to the compacting stage or the transfer method.
- Relocate the injection points so that the flow fronts, which form the welding lines, form at higher temperatures and/or may continue to flow for a better mixture. Use injection simulation programs in order to optimize the relocation of the injection points.
- Increase the thickness of the piece in the welding line.
- Increase the wall thickness of the piece.
- Use a textured mold surface in order to hide the welding line.

Contamination:

- Examine the handling or transport system of the raw material.
- Purge the barrel.
- Clean and check the hopper, the magnets, the feeder and the lines that feed the injector. Clean the workplace in order to avoid contamination.
- Check and/or clean the gun molding mill blades.
- Clean the ejector pins and ventilation of the mold.
- Use anti-static to avoid superficial contamination with dust and others.
- Change the lot of the material.

Deformed or long parts due to difficult molding:

- Increase the cooling time.
- Use an unmolding machine.
- Decrease the compacting time or pressure.
- Lower the mold temperature by half where the ejectors are located and/or increase the one on the other half.
- Decrease the initial opening speed of the mold.
- Level route of the ejector pins and/or decrease its speed.
- Check the mold looking for grooves and cracks. Improve the finish of the mold (polish).
- Increase the inclination angle in the cavity in order to facilitate the ejection of the part.
- Reduce the bas-relief and remove the bulges or irregularities or rough edges of the mold.
- Change the number of touch of the ejector pins.
- Increase the size or amount of ejector pins.

Bending or curving of the part:

- Modify the cooling of the halves of the mold in order that the part may cool in an even manner. Check the possible obstructions in the cooling system.
- Increase the capacity or efficiency of the cooling system.
- Check the cavity looking for hot spots.
- Increase the cooling time.
- Decrease the mattress in the polymer dose.
- Reduce the time of the front screw.
- Submerge the piece in water in order to help the cooling.
- Adjust the compacting pressure, together with the injection speed.
- Use retainers or guides to avoid deformation of the piece while it is being cooled outside the mold.
- Check the plug in order to verify if bending or shifting has occurred.
- Decrease the volumetric dose in order to limit the packing of the material in the part.
- Relocate the cavity entry.
- If the bending or curving of the part only occurs with some formulations, change the pigment or the load being used, enlarge or change the mold entry position, use a resin with a narrower molecular weight distribution.
- Redesign the piece in order to obtain a more homogeneous wall thickness or one with gradual changes.
- Verify that the number of ejectors is sufficient for the size of the piece.
- Use injection simulation programs for improving the cooling system.

Dimensional instability (very small parts – excessive contraction):

- Increase the compacting pressure.
- Increase the injection speed.
- Adjust the time of the front screw, verify the sealing time of the cavity entry.
- Make sure that the material mattress is maintained; otherwise, adjust the size of the shot and/or the reverse pressure. Check the anti-return valve.
- Increase the closed mold time: compacting or cooling.
- Decrease the mold temperature. Check whether there are heat concentration points in the mold.

TROUBLESHOOTING GUIDE FOR INJECTION MOLDING



- Reduce the room temperature or use cooling implements in the post-molding.
- Increase the entry size to the cavity.

Dimensional instability (very large or heavy parts):

- Decrease the cast material mattress and the time of the front screw.
- Increase the water cooling temperature.
- Check if the part shows irregularities or rough edges.
- Decrease the necessary time for the ejection of the part.

Dimensional instability (variability in the weight or dimensions of the piece):

- Check whether there are fluctuations in the return time of the screw. In order to counteract this, change the rotation speed of the screw (rpm), check the cooling of the hopper, verify that the granules (pellets) of the virgin and recovered resin are of equal size or change the profile of the temperatures in the barrel.
- Check if there are fluctuations in the cast material mattress. In order to counteract this, check the proper operation of the anti-return valve, increase the injection speed, increase the reverse pressure (counter-pressure) or optimize the time of the front screw. Check the consistency of the recycled material.
- Analyze the changes in room temperature, if there are any variations in the temperature controllers of the barrel and in the oil temperature.
- Check the nozzle in order to verify that there are no partial obstructions; increase the heating in the nozzle.
- Verify that the mold temperature is homogeneous. Check whether the coolant temperature is constant, check the entries and exits of the coolant, adjust the flow and the temperature of the coolant and/or open the cooling ducts.
- Make sure that the reverse pressure (counter-pressure) is homogeneous. Readjust the counter-pressure, check the settlings between the nozzle and the mold and/or check for wear of the screw.
- Check if the speed of the screw is constant. Check the hydraulic system, check the electric

motor power transmission or change the profile of the temperatures.

Fragile parts:

- Verify that the casting temperature is suitable.
- Reduce the speed of the screw and or the reverse pressure.
- Verify that the raw material is not contaminated.
- Check the dispersion of the additives, pigments and recycling.
- Check the part looking for superficial defects, contamination, bending or curving of the part and make the necessary corrections.
- Check the uniformity of the mold cavities.
- Improve the design of the injected piece: locating the injection points, thickness of the wall, nerves, etc. Use injection simulation programs for improving the design.
- Use polyethylene of greater molecular weight.

TROUBLESHOOTING GUIDE FOR INJECTION MOLDING



References:

- Advanced Process Engineering. Injection Molding Troubleshooting Guide, second edition. Phoenix, Arizona, USA.
- Moreno, Hector. Guia para la Resolucion de Problemas Moldeo por Inyeccion. Gerencia Comercial – Departamento de Servicios Tecnicos y Desarrollo. 1999.

This bulletin has been made by the Marketing Department of Polinter with the support of the specialists of Investigación y Desarrollo, C.A. (INDESCA) and by the Technical Services Department of CORAMER. This is intended for all clients and users of the Venelene® resins and we trust that the information contained herein is helpful and useful.

Please contact us at the following email address, info@polinter.com.ve or through our agent: Corporacion Americana de Resinas (CORAMER), with branch offices in Venezuela, Colombia, Peru, Ecuador and Chile (<http://www.coramer.com>), should you have any suggestions or comments regarding this issue.

The information described in this document is, to our best knowledge, accurate and truthful. However, since the particular uses and transformation conditions are completely out of our hands, the adjustment of the parameters in order to reach the maximum performance of our products for a specific application depends on and is the responsibility of the user.

In order to obtain more detailed information of the security aspects regarding the use and disposal of our products we invite you to consult the security pages (MSDS) of the Venelene® polyethylene.