TROUBLESHOOTING GUIDE FOR INJECTION MOLDING

Table Of Contents

Black Specks

Black specks are non miscible particles only seen on the surface of an opaque part because pigments tend to hide them. With transparent parts they are visible throughout.

Cause	Solution
Machine Contamination	Increase melt temperatures to help remove degraded resin. Purge well with thermally stable resin or purging compound. Remove screw and clean and inspect screw, screw tip & check ring for dead spots (i.e. hang up areas).
Burning Of Resin	Lower melt temperatures, injection speeds, screw speed and back pressure to reduce both shear input and thermal input. Verify that heater bands are not overriding.
Mold Contamination	Clean the mold surface. Ensure that there is no plate out or residue building up. Wipe ejector pins and cams of oils and greases.
Resin Contamination	Examine virgin, regrind & concentrates for foreign material. Look for black specks tumbled with pellets as well as within the pellets and regrind. Check grinders, dryers and hoppers.

Blisters

Blisters are areas of trapped gas seen as surface irregularities or bumps on the surface of the molded part. They occur during injection or cooling.

Cause	Solution
Large Regrind Particles	Decrease particle size of regrind by decreasing the blade gap and/or screen mesh. This provides a homogeneous melt without air entrapment.
High Screw Speed	Decrease the screw rpm's in order to provide a compressed melt state. This will allow even feed of pellets into the flights of the screw with less air entrapment.
Poor Venting	Increase amount of vents and the size of the vents. Add venting where needed. This will help any trapped air to escape. Add runner venting where necessary.
Low Back Pressure	Increase the back pressure to compress the melt. This will force any trapped air out.
Poor Gate Location	Relocate gate to minimize any flow disruptions.
Low Mold Temperature	Raise mold temperature to slow the cooling of the polymer. This will allow for more sufficient packing of the resin and more time before gate freeze-off.

Blush

Blush often occurs near the gate or a flow disruptor such as a core pin or shutoff. It is also noticed near sharp corners or severe wall thickness transitions. It is the result of melt fracture. The part surface will appear dull (low gloss) or discolored and some flow marks will be visible.

Cause	Solution
High Injection Velocity	Lower entire injection speed. This will impart less shear and damage to the material at the gate. Profile injection speed and inject slowly as resin first approaches the gate.
Cold Slug	Increase the size of the cold slug well opposite the sprue and at the end of runners. This will gather and hold and semi-solid material that would cause a gate obstruction.
Low Injection Pressure	Increase injection pressure in order to better pack out the part and force the resin to reflect the tool surface.
Sharp Corners	Radius as generously as possible any sharp corner or severe geometry change. Smooth out wall thickness transitions. Allow material to flow gently through the cavity.
Small Gate	Increase gate size to allow for a gentle transition into the cavity.
Other Process Changes	Increase melt & mold temperature.

Brittleness

Brittleness occurs from degrading the resin and results in a reduction of physical properties. Brittleness can also be related to moisture in some materials prior to melt processing.

Cause	Solution
High Melt Temperature	Reduce melt temperature to prevent thermal degradation.
Long Residence Time	Use smaller barrel capacity or speed up cycle. Excessive residence time in the barrel can have the same effect as too high a temperature. Ensure a shot size to barrel capacity ratio of 40-80%.
Moisture	Some hygroscopic resins become brittle if not properly dried. Ensure Dew points are –20º F or better and there is adequate air throughout. Check moisture level with analyzer.
Over Packing	Decrease hold pressure, time, & shot size. By forcing too much material in to the tool, it creates molded in stresses, especially near gates. These areas break easily.
Shear Degradation	Reduce screw speeds, back pressure, & injection velocity.
Low Mold Temperature	Raise mold temperature or raise melt temperature to minimize required pressure to fill and, thus reduce molded in stress.

Bubbles

Regions within the molded part without plastic are referred to as internal voids or bubbles. They can be caused by differential cooling or air entrapment.

Cause	Solution
Insufficient Venting	Increase the size of the vents, the amount of vents or relocate vents. Clean vents thoroughly.
Moisture	Increase drying time, drying temperature and/or lower the dew point. Measure resin moisture level.
High Melt Temperature	Lower melt temperature to help reduce the cooling cycle and minimize material shrinkage.
Fast Cooling	Increase mold temperature to reduce differential cooling.
Under Packing	Increase injection & hold pressures and/or times.
Large Cushion	Decrease cushion to allow for better transfer of pressure to the cavity.
Small Gate/ Poor Location	Increase both gate and runner sizes to ensure they do not freeze off before the cavity is completely packed out. Relocate gate.
Part Design	Eliminate excessive wall thickness or variations in wall thickness.

Burns

Burned resin can be the result of several things. High barrel temperatures, shear from screws or runner and gate systems, excessive residence time and poor venting will all cause the resin to discolor.

Cause	Solution
High Injection Velocity	Lowering the injection speed will allow trapped air in the mold to escape. It will also reduce shear heating of the resin.
Insufficient Venting	Increase the size, amount and/or location of the vents.
Excessive Residence Time	Reduce time the resin resides in barrel by speeding cycle or decreasing barrel size.
Excessive Melt Temperature	Lower barrel heaters, screw speed and back pressure to reduce resin temp.
Wet Material	Properly dry hygroscopic resin.
Damaged Heater Band	Check that heater bands are not overriding or burnt out.
Dead Zones	Ensure that no dead zones are in resins path. These hang-up areas will collect and degrade material that breaks free on injection.
Aggressive Screw	Ensure a low compression screw with adequate flight depths & a short metering section for shear sensitive materials.
Small Gates	Excessively small gates and runners will shear and overheat the resin. Increase diameters to reduce this.

Cloudiness/Haze

Cloudiness in transparent resins can be the result of contamination. Other causes can be mold surfaces, mold temperature or resin temperature.

Cause	Solution
Resin Contaminated	Run only virgin to see if problem improves. Clean dryer, hoses, hopper and purge barrel.
Wet Material	Verify all drying conditions for resin and ensure moisture level is within recommended range.
Over Packing	Reduce hold pressure and/or time to eliminate cloudiness in gate region.
Under Packing	Increase holding pressure and/or time as well as melt & mold temperature to better reflect tool surface.
Over Heating	Some opaque resins will appear hazy when overheated. Lower melt temperature, screw speed and back pressure.
Mold Surface	Dull mold surface or cold mold surface. Ensure there is no residue coating to mold. Raise temperature to ensure good packing.
Release Agents	Spray on mold release may build up and prevent accurate reflectance of the mold surface. Wipe clean and examine for plate out.

Contamination

Contaminants often show up as black specks or color streaks.

Cause	Solution
Foreign Materials	Review resin handling system to ensure no foreign materials are mixed with virgin or regrind.
	Switch to all virgin to isolate source.
	Purge barrel and clean hopper, dryer and hose lines.
	Clean grinders, filters and containers.
	Ensure magnet is in place and no objects are attached to it.

Cracking/Crazing

Cracking and crazing can occur upon ejection from the mold. Many simple changes will improve the part quality after molding.

Cause	Solution
Thick Wall Section	High shrinkage or low molecular weight resin may tend to crack or generate voids upon cooling. Using a higher molecular weight or lower shrink resin will help eliminate this. Thin wall sections will tend to shrink less.
Stress	Molded in stress can make a part brittle and cause cracking. Lowering the injection velocity and increasing the mold temperature will improve this problem.
Rapid Cooling	Quenching will impart brittleness and cracks on many resins. Increase the mold temperature and lengthen the cooling time.
Wet Material	Many hygroscopic resins will become brittle and crack on ejection if processed wet. Verify that all drying parameters are properly set for the given resin.
Under Packing	Increase the injection forward time, injection velocity and packing phase to ensure the part is full and weld lines are strong.
Chemical Attack	Verify mold release and cleaners are compatible with the base resin.
Small Gate	Enlarge gate to reduce stresses.

Deformation

Upon ejection, parts can stick to the tool causing the geometry to be deformed as the ejectors push forward. Deformation can also be caused on mold opening if the part tries to stick to the wrong half.

Cause	Solution
Part Not Cured	Increase cooling time, decrease mold temperature and lower melt temperature so that the part will set up quicker.
Over Packing	Decrease injection and holding speeds and pressures to reduce the packing of the cavity.
Low Draft Angles	Increase draft on the tool's core, cavity and standing features to eliminate undercuts. Ensure that textured surfaces have enough draft.
Excessive Shrink	Increase packing of the part to maximize material input to the cavity and reduce shrinking to the core.
Differential Cooling	Increase tool temperature on ejector half to induce more shrinkage to that side.
No Lubricant	Ask for additional lubricant on resin from material supplier or add mold coatings.
Fast Ejection	Lower ejection rate to allow part to peel off or bend over undercuts.

Delamination

Delamination is a separation of layers of the molded part. These layers can develop from contamination or improper processing.

Cause	Solution
Contamination	Review resin handling system to ensure no foreign materials are mixed with virgin or regrind. Switch to all virgin to isolate source. Purge barrel and clean hopper, dryer and hose lines. Clean grinders, filters and containers. Ensure magnet is in place and no objects are attached to it.
Non-Miscible Resin	Cross contamination with other resins or separation of alloys and blends can cause delamination. Ensure process is set properly for blends & alloys and look for contamination.
Sharp Corners	Round all sharp corners or wall thickness changes. This will propagate laminar flow.
Uneven Melt Temperature	Improve homogeneity of melt by increasing screw speed, back pressure or compression ratio of screw. This will provide more shear heating and mixing of melt.
Incorrect Mold Temperature	Adjust the mold temperature to the suggested range for the given resin.
Fast Injections Speed	Slow injection speed to reduce shearing and polymer separation.

Dimensional Instability

Dimensional instability or warpage in a molded part can be the result of differential material shrinkage, molded-in-stress relief, or an inconsistent injection molding process.

Cause	Solution
Under Packing	Increase pressures and times to better pack out the parts.
High Melt Temperature	Decrease melt temperature and increase cooling time to minimize the amount of shrinkage that occurs.
Inconsistent Shot Size	Examine check ring or non-return valve for damage/improper seating.
No Cushion	Increase cushion to help transfer packing pressure to the part.
Gate Freeze-Off	Increase gate size or decrease land length to increase the amount of time there is to fill and pack.
Warm Mold	Decrease the mold temperature to cool part quicker and minimize shrinkage.
Unbalanced Runner	Adjust volumes of runner system so that all cavities fill at the same rate.
Non-Uniform Cooling	Check line layout & measure each half’s temperature & water throughput.
Over Packing	Back off second stage time & pressure to reduce molded in stress.

Discoloration

Discoloration can be caused by several factors and can occur to the actual pellets prior to molding as well as the final part.

Cause	Solution
Contamination	Purge out injection barrel until resin runs clean.
Burning	Lower melt temperature, screw speed and/or back pressure. Also, decrease injection velocity.
Long Residence Time	Reduce cycle time, use small volume barrel. Use a reverse temperature profile.
Poor Throat Cooling	Verify feed throat temperatures and reduce feed zone temperature.
Not Enough Heat Soak	Shot capacity is too small, move to large barrel volume to aid dispersion of additives and reduce severe shear heating.
Heater Band Override	Too much shear from screw. Verify proper screw design, raise rear zone temperatures to lower the viscosity of the resin and shear effects.

Flash

Flash is caused when resin flows out of the cavity at the parting line. This can be a function of many machine, mold and material variations and is typically easily corrected.

Cause	Solution
Exceeding Clamp Pressure	Increase clamp pressure set point. Lower pack and hold pressures. Lower injection velocity.
Too Much Volume	Reduce the shot size and/or increase the cushion.
Low Resin Viscosity	Lower melt temperature to prevent easy flow into vents and parting line.
Poor Drying	Increase drying of hygroscopic resins. Moisture can reduce molecular weight and increase flow.
Solid Skin Not Cured	Begin packing with very low pressure to allow skin to set at points of flash and then ramp up the packing and holding pressures.
Damaged Parting Line	Inspect and repair worn or damaged regions of the parting line and vents.
Vents Too Deep	Verify the proper vent depth for the material.

Flow Lines

Flow lines, weld lines, meld lines, knit lines and ripples or folds are all a result of a disturbance in the flow path of the resin. When smooth laminar flow is interrupted, the polymer may not accurately reflect the surface of the tool.

Cause		Solution
Low Melt Temperature	→	Increase melt temperature to aid flow, packing and improve reflection of tool surface.
Poor Gate Location	→	Locate gate to fill thickest section first and produce the smallest amount of weld lines.
Low Mold Temperature	→	Raise mold temperature so skin does not set up as quickly and packing phase is effective in replicating mold surface.
Cold Material Entering Cavity	→	Ensure the cold slug wells are is large enough to catch any frozen material from the nozzle. Add wells at the end of runners.
Slow Injection Velocity	→	Inject fast to push welds together while material is hottest.
Clogged Vents	→	Clean vents and verify depth is correct for the given material.
Under Packed	→	Increase packing pressure and time to push flow front together.

Jetting

Jetting or worming is caused by excessive injection force without a barrier for the melt to hit. The material cannot develop a laminar flow front unless it impinges on a wall or boss.

Cause	Solution
Fast Injection Velocity	Lower injection speeds or profile injection with slow initial speed, then ramp it up.
Small Gate	Increase gate size to reduce nozzle effects.
Gate Location	Move gate so that material impinges on a wall or boss immediately upon entering the cavity.
Low Melt Temperature	Raise melt temperature to reduce viscosity and aid formation of a flow front.
Gate Land Length	Reduce the land length of the gate. Review material supplier recommendations for this dimension based on specific material.
Nozzle Freeze Off	Raise nozzle temperature to ensure material is not solidifying.
Runner Design	Incorporate an “S” configuration runner so that flow has to change direction upon entering the cavity.

Mold Deposits

Under the high temperatures and pressures of the injection molding process, additives and residual monomer can be volatized and separated from the resin. These materials can end up as residue on the mold surface and require cleaning or leave surface defects on the molded parts.

Cause	Solution
Over Heating	Reduce melt temperature of the resin, lower back pressure and/or screw speed.
Long Residence Time	Reduce residence time of resin in the barrel by running a faster cycle or moving to a press with smaller shot capacity.
Shear Heating	Lower injection velocities, verify that gate sizes are proper and increase as necessary.
Poor Venting	Verify vent sizes are correct for the given resin. Increase the number of vents at more locations.
Mold Temperature	Verify that mold surfaces are set at the material suppliers recommended range. Too cold or too hot of a surface can increase deposits.
Wet Material	Ensure that the resin has been properly dried. Moisture, under temperature and pressure can provide a route for volatiles to escape from the resin.

Short Shots

A short shot occurs when insufficient material is injected and does not completely fill or pack out all of the cavities in the mold. Shorts can be caused by machine, mold or material.

Cause	Solution
Insufficient Melt Volume	Adjust shot size and cushion size until part is properly filled.
Low Injection Velocity	Raise injection velocity to push material into cavity before gate freeze off. Lower injection velocity if air is being trapped at the end of fill.
Low Injection Pressure	Raise injection pressure.
Low Melt Temperature	Increase melt temperature to ease injection. Raise barrel set points, screw speed and/or back pressure.
Low Mold Temperature	Increase mold temperature to aid flow and prevent too thick of a solid skin from setting.
Poor Fill Profile	Ensure that 95% of cavity is filled by 1st stage injection and the remainder is filled and packed under 2nd stage (hold).
Air Trap	Clean vents and ensure they are the proper size. Burning may also accompany this.
Low Back Pressure	Increase the back pressure so that adequate melt densification occurs.
Damaged Barrel/Screw	Verify clearances to prevent back flow.

Sink Marks

Sink marks can be the result of a poor fill (short shot) or the result of differential cooling when wall sections vary greatly in thickness.

Cause	Solution
Insufficient Feed	Increase shot size and/or decrease cushion to push more resin into the cavity. Ensure consistent cushion size.
Insufficient Pressure	Increase the screw forward time under pressures to ensure gate freeze off is obtained.
Cold Mold	Slow the cooling/increase mold temp so that the outer skin and the interior of the part shrink at the same rate.
High Mold Temperature	Decrease mold temperature on the side of the sink to thicken the frozen layer and retard the shrinkage around that sink mark.
High Melt Temperature	Lower the melt temperature to, in effect, lower the amount of heat that needs to be pulled from the part. This will help reduce differential cooling and shrinkage.
Gate Location	Ensure gating is from thick to thin to achieve the best filling and packing possible.
Runner & Gate Size	Increase runner and gate sizes to get best pressure transfer to the part in the cavity of the mold.
Poor Part Design	Use nominal wall design principles with standing features at 50% of nominal wall thickness.

Splay

Three types of splay are often encountered in injection molding. Splay due to moisture usually results in silver streaks on the surface and slight delaminating of a film layer. Heat splay, caused by degradation or hang-up, looks similar but is often accompanied by black specks or yellowness. Splay due to contamination is often accompanied by severe delamination.

Cause	Solution
Moisture	Check dryer's dew point, temperature and residence time. Check hoses for holes.
High Melt Temperature	Verify barrel set points and measure purge shot with a pyrometer. Lower if necessary.
Shear	Lower injection velocity and screw speeds. Inspect nozzle, sprue, runners and gates for hang up points and sharp corners. Ensure that screw design is not too aggressive.
Decompression	Turn decompression off.
Clogged Vents	Clean and inspect all vents. Verify venting is the proper depth for the given resin.
Contamination	Inspect virgin and regrind for possible contaminants. Try running just virgin from another package.
Condensation	Increase mold surface temperature.

Surface Finish

A dull surface finish or poor reflection of mold surface can be easily corrected.

Cause	Solution
Cold Material	Raise melt temperature to decrease resin viscosity and allow easier packing. This will better reflect the mold surface.
Air Trap	Add venting or increase vent size at each point of air entrapment.
Cold Mold	Raise mold temperature to allow easier flow and allow resin to reflect surface finish better.
High Melt Temperature	Lower melt temperature to prevent out gassing and residue build up in cavity.
High Injection Velocity	Slow injection speed to allow air and volatiles to escape rather than get caught in the flow front.
Moisture	Ensure resin has been properly dried. Verify dew point, temperature and residence time in the dryer.
Contamination	If clear parts appear cloudy, try running virgin from a new package to eliminate any possible contaminants in regrind, drying or from handling.
Mold Surface	Inspect the mold surface for deposits, residue or contaminants.

Voids/Bubbles

Internal voids on a part are simply gaps or pockets with no material. The are often invisible but can greatly reduce the strength. These are caused by under packing or shrinkage.

Cause	Solution
Low Pressure	Increase injection, hold and pack pressures and times to ensure the cavity is completely filled through gate freeze off, monitor part weight until it plateaus.
Insufficient Volume	Increase shot size and/or lower cushion to put more volume of rein in to cavity.
Premature Freeze Off	Increase size of nozzle, sprue, runners and gates to allow for longer packing.
Low Mold Temperature	Raise mold temperature to allow for slower cooling of skin. This will make material inside less likely to pull away from itself.
High Melt Temperature	Lower melt temperature to ensure part cures as fast as possible with minimal shrinkage.
Thick Wall Sections	Design part with uniform wall thickness. Ensure ribs are on 50% of nominal wall thickness.

GENERAL POLYMER PROCESSING GUIDE INTRO

Polymer families tend to have a wide range of processing conditions so supplier specific information and grade specific information should be used whenever possible when designing molds or setting up processing conditions.

The drying conditions, melt and mold temperature ranges provided should only be used as a general guide. Because the specific processing conditions can vary with different grades of a specific material as well as from supplier-to-supplier it is strongly suggested to refer to the suppliers data sheet for information specific to a given grade of material.

The mold shrinkage values provided are general ranges and are only intended to be used to allow comparisons to other materials and should only be used as a general guide. The mold shrinkage values provided are the “flow-direction” shrinkage and are based on 1/8” thick injection molded test specimens tested per ASTM D955. Actual material shrinkage is based on a number of factors including part design, wall thickness, tool configuration, mold cooling layout and processing parameters. Entec’s recommendation would be to use shrinkage values observed in other molds currently running the material that produce parts with a similar geometry and wall thickness, and proceed to machine the mold cores and cavities in a “steel safe” manner.

GENERAL POLYMER PROCESSING GUIDE

Resin	Specific Gravity	Drying Parameters Temperature (F)/Time	Mold Temperature (F)	Melt Temperature (F)	Mold Shrinkage (in./in.)
ABS Flame Retardant	1.18	180 / 3 hrs	105 - 160	345 - 365	.003 - .007
ABS/TPU	1.1	170 / 4 hrs	80 - 100	390 - 415	.005 - .007
ASA	1.07	175 / 2-4 hrs	105 - 175	465 - 535	.004 - .007
EVA	.920 - .970	None Required	60 - 105	300 - 425	.001 - .016
GPPS	1.04	170 / 2 hrs	60 - 160	390 - 475	.003 - .007
HIPS	1.05	170 / 2 hrs	60 - 160	390 - 475	.003 - .007
LCP Reinforced	1.50 - 1.90	250 - 300 / 4 hrs	175 - 250	555 - 650	.000 - .004
PA 6 (Nylon 6)	1.13	165 / 2-4 hrs	160 - 200	460 - 520	.010 - .015
PA 6 Reinforced	1.18 - 1.49	165 / 2-4 hrs	160 - 220	515 - 565	.0015 - .003
PA 6/6 (Nylon 6/6)	1.14	165 / 2-4 hrs	175 - 200	520 - 530	.012 - .020
PA 6/6 Reinforced	1.22 - 1.49	165 / 2-4 hrs	175 - 220	540 - 570	.003 - .005
PBT	1.31	250 / 3-4 hrs	100 - 200	460 - 500	.017 - .023
PBT Reinforced	1.52	250 / 3-4 hrs	140 - 220	480 - 525	.003 - .006
PC	1.2	250 / 4 hrs	160 -200	550 - 600	.005 - .007
PC Reinforced	1.25 - 1.52	250 / 6 hrs	190 - 250	600 - 650	.001 - .005
PC/ABS	1.08 - 1.22	250 / 3 hrs	150 - 190	460 - 500	.005 - .007
PE	.905 - .968	None Required	85 - 105	320 - 450	.015 - .035
PET Reinforced	1.58 - 1.73	250 / 3 hrs	180 - 250	540 - 580	.002 - .006
PMMA (Acrylic)	1.19	170 / 3 hrs	85 - 160	350 - 450	.004 - .006
POM (Acetal)	1.41	180 / 1 hr	170 - 200	370 - 390	.015 - .022
PP	0.9	None Required	80 - 150	375 - 500	.010 - .025
PPO	1.04 - 1.28	220 - 250 / 6 hrs	160 - 220	550 - 620	.004 - .007
PPS Reinforced	1.4 - 2.0	265 - 285 / 3-4 hrs	285 - 320	560 - 650	.002 - .007
SAN	1.06	175 / 3 hrs	105 - 175	390 - 480	.003 - .007
TPU	1.01 - 1.23	160 - 220 / 3 hrs	50 - 110	365 - 435	.005 - .01

The information presented in this document was assembled from literature of the resin product producer(s). The information is believed to be accurate however Entec Polymers ("Entec") makes no representations as to its accuracy and assumes no obligation or liability for the information, including without limitation its content, any advice given, or the results obtained. ENTEC DISCLAIMS ALL WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING FITNESS FOR A PARTICULAR PURPOSE. The customer shall use its own independent skill and expertise in the evaluation of the resin. product to determine suitability for a particular application and accepts the results at its sole risk.

ENTEC POLYMERS

1900 Summit Tower Blvd., Suite 900

Orlando, FL 32810

EntecPolymers.com

833.609.5703