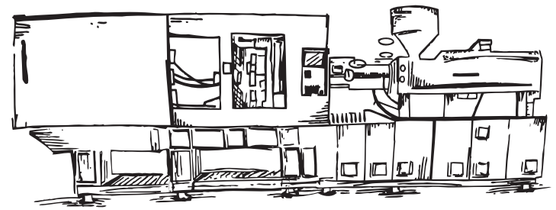
Polymer	Specific Gravity	Drying Parameters: Temperature (F)/Time	Max Moisture Content for Injection Molding (%)	Mold Temperature (F)	Melt Temperature (F)	Mold Shrinkage (in)	Rule of Thumb Tonnage (Tons per Square Inch)	Vent Depth (in)	Vent Land (in)
ABS	1.04	180/3 hours	0.10	120-140	440-465	0.003-0.007	2.5-3.5	0.0015-0.0025	0.030-0.050
ABS + Nylon	1.07-1.12	175-190/4 hours	0.10	105-175	465-520	0.008-0.009	3.0-4.0	0.0005-0.0010	0.020-0.040
ASA	1.07	175/2-4 hours	0.04	105-175	465-535	0.004-0.007	2.5-3.5	0.0015-0.0025	0.030-0.050
CoPolyester	1.17-1.19	180-190/4-6 hours	0.02	100-150	500-540	0.005-0.007	3.0-5.0	0.0005-0.0015	0.040-0.060
EVA	0.92-0.97	Not Required	NH	60-105	300-425	0.007-0.020	2.0-3.0	0.0005-0.0007	0.040-0.060
GPPS	1.04	170/2 hours	NH	60-160	390-475	0.003-0.007	2.0-2.5	0.0010-0.0030	0.030-0.050
HIPS	1.05	170/2 hours	NH	60-160	390-475	0.003-0.007	2.0-2.5	0.0010-0.0030	0.030-0.050
LCP	1.50-1.90	250-300/4 hours	0.01	175-250	550-650	0.000-0.004	1.5-2.5	0.0005-0.0010	0.020-0.040
PA6 (Nylon 6)	1.13	165/2-4 hours	0.20	180-200	460-520	0.009-0.012	3.0-4.0	0.0005-0.0010	0.020-0.040
PA6 Reinforced	1.18-1.49	165/2-4 hours	0.12	180-220	515-565	0.002-0.005	4.0-5.0	0.0005-0.0025	0.030-0.050
PA 66 (Nylon 66)	1.14	165/2-4 hours	0.20	150-200	520-530	0.015-0.020	3.0-4.0	0.0005-0.0010	0.020-0.040
PA66 Reinforced	1.22-1.49	165/2-4 hours	0.12	180-220	540-570	0.003-0.008	4.0-5.0	0.0005-0.0025	0.030-0.050
PBT	1.31	280/3-4 hours	0.02	100-200	460-500	0.017-0.023	3.0-4.0	0.0005-0.0015	0.030-0.050
PBT Reinforced	1.52	280/3-4 hours	0.02	140-220	480-525	0.003-0.006	4.0-5.0	0.0005-0.0020	0.030-0.040
PC	1.2	250/4 hours	0.02	160-200	550-600	0.005-0.007	4.0-5.0	0.0010-0.0030	0.030-0.050
PC Reinforced	1.25-1.52	250/6 hours	0.02	190-250	600-650	0.001-0.004	4.0-5.0	0.0010-0.0030	0.030-0.050
PC/ABS	1.08-1.22	250/3 hours	0.03	150-190	460-500	0.005-0.007	3.0-4.0	0.0010-0.0030	0.030-0.050
PC/PBT	1.19-1.24	250/3-4 hours	0.02	130-180	490-520	0.007-0.016	4.0-5.0	0.0005-0.0010	0.030-0.060
PE-HDPE	0.940-0.965	Not Required	NH	50-200	400-500	0.025-0.035	2.5-3.5	0.0008-0.0010	0.020-0.040
PE-LDPE	0.915-0.935	Not Required	NH	70-140	400-500	0.015-0.026	2.0-3.0	0.0005-0.0007	0.020-0.040
PE-LLDPE	.915-.930	Not Required	NH	70-160	400-500	0.015-0.035	2.0-3.0	0.0005-0.0007	0.020-0.040
PET Reinforced	1.44-1.73	250/3 hours	0.02	180-250	540-580	0.002-0.005	4.0-5.0	0.0005-0.0010	0.030-0.060
PMMA (Acrylic)	1.19	170/3 hours	0.06	85-160	350-450	0.002-0.006	3.0-4.0	0.0015-0.0030	0.040-0.050
POM (Acetal)	1.41	180/1 hour	NH	170-200	370-390	0.018-0.035	3.0-4.0	0.0005-0.0015	0.040-0.050
PP	0.9	Not Required	NH	100-210	460-520	0.010-0.025	2.5-3.5	0.0010-0.0020	0.020-0.040
PPS	1.40-2.00	260/3-4 hours	0.06	275-325	560-640	0.003-0.010	3.5-4.5	0.0005-0.0010	0.030-0.050
PSU	1.30-1.37	260-300/4 hours	0.02	280-350	640-730	0.008-0.010	4.0-5.0	0.0010-0.0015	0.030-0.050
SAN	1.06	175/3 hours	0.10	105-175	390-480	0.002-0.005	2.5-3.5	0.0015-0.0025	0.030-0.050
TPE	0.87-1.20	180/3 hours	0.10	70-100	370-460	0.017-0.047	2.5-3.5	0.0008-0.0010	0.100-0.150
TPU	1.01-1.23	160-220/3 hours	0.02	50-110	365-435	0.012-0.17	3.0-4.0	0.0005-0.0010	0.030-0.060
TPV	0.92-0.97	180/3 hours	0.10	50-125	380-450	0.010-0.050	2.5-3.5	0.0008-0.0010	0.100-0.150

While we do everything possible to ensure accuracy of all information found on our chart, Absolute cannot assume responsibility for any errors or oversights. We therefore recommend contacting your material supplier to confirm the accuracy of any information shown.

HAITIAN
ABSOLUTE



PROCESSING GUIDELINES

LEADERSHIP
HAS MANY DIMENSIONS
communication. innovation. efficiency.

Absolute Haitian

94 Gardner Street, Worcester, MA 01610
Tel: 508-459-5372 • Fax: 508-459-5368

Ohio Tech Center, 12900 Plaza Drive, Parma, OH 44130
Tel: 216-452-1000 • Fax: 216-767-5084

www.absolutehaitian.com

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.

- Increase melt temperatures to help remove degraded resin.
- Purge well with thermally stable resin or purging compound.
- Remove screw, clean and inspect screw, screw tip and check ring for dead spots (i.e. hang up areas).
- Lower melt temperatures, injection speeds, screw speed and back pressure to reduce shear and thermal input.
- Verify that heater bands are not overheating.
- Clean the mold surface.
- Examine virgin, regrind and concentrates for foreign material.

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.

- Decrease particle size or regrind by decreasing the blade gap and/or screen mesh.
- Decrease the screw Rpm's in order to provide a compressed melt state.
- Increase amount of vents and the size of the vents, add venting where needed.
- Increase the back pressure to compress the melt.
- Relocate gate to minimize any flow disruptions.
- Raise mold temperature to slow cooling polymer.

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 the melt fracture. The part surface will appear dull (low gloss) or discolored and some flow marks will be visible.

- Lower entire injection speed.
- Increase the size of the cold slug well opposite the sprue and at the end of runners.
- Increase injection pressure.
- Radius as generously as possible any sharp corner or severe geometry change, smooth out wall thickness transitions.
- Increase gate size to allow for a gentle transition into the cavity.
- Increase melt and 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.

- Reduce melt temperature.
- Use smaller barrel capacity or speed up cycle. Ensure a shot size to barrel capacity ratio is 40-80%.
- Some hygroscopic resins become brittle if not properly dried. Check moisture level with analyzer.
- Decrease hold pressure, time and shot size.
- Reduce screw speeds, back pressure and injection velocity.
- Raise mold temperature or raise melt temperature to minimize required pressure to fill and thus reduce molded stress.

BURNS

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

- Lowering the injection speed will allow trapped air in the mold to escape. It will also reduce shear heating of the resin.
- Increase the size, amount and/or location of the vents.
- Reduce time the resin resides in barrel by speeding cycle or decreasing barrel size.
- Lower barrel heaters, screw speed and back pressure to reduce resin temperature.
- Properly dry hygroscopic resin.
- Check that heater bands are not overriding or burnt out.
- Ensure that no dead zones are in resin path. These hang-up areas will collect and degrade material that breaks free on injection.
- Ensure a low compression screw with adequate flight depths and a short metering section for shear sensitive materials.
- Excessively small gates and runners will shear and overheat the resin. Increase diameters to reduce the effects.

BUBBLES

Regions within the molded part without plastic are referred to as internal voids or bubbles. Differential cooling or air entrapment can cause them.

- Increase size of the vents, the number of vents or relocate vents. Clean vents thoroughly.
- Increase drying time, drying temperature and/or lower dew point. Measure resin moisture level.
- Lower melt temperature to help reduce differential cooling.
- Increase injection and hold pressures and/or times.
- Decrease cushion to allow for better transfer of pressure to the cavity.
- Increase both, gate and runner sizes, to ensure they do not freeze off before the cavity is completely packed out. Relocate gate.
- Eliminate excessive wall thickness or variations in wall thickness.

CLOUDINESS / HAZE

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

- Run only virgin to see if it solves the problem. Clean dryer, hoses, hopper and purge barrel.
- Verify all drying conditions for resin and ensure moisture level is within recommended range.
- Reduce hold pressure and/or time to eliminate cloudiness in gate region.
- Increase holding pressure and/or time, as well as, melt and mold temperature to better reflect tool surface.
- Some opaque resins will appear hazy when overheated. Lower melt temperature, screw speed and back pressure.
- Dull mold surface or cold mold surface. Ensure there is no residue coating to mold. Raise temperature to ensure good packing.
- Spray on mold release may build and prevent accurate reflectance of the mold surface. Wipe clean.

CONTAMINATION

Contaminants often show up as black specks or color streaks.

- Review resin-handling system to ensure no foreign materials are mixed with virgin or regrind.
- Switch to all virgin or regrind.
- 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.

- High shrinkage or low molecular weight resin may tend to crack or generate voids upon cooling. Using a higher molecule weight or lower shrink resin will help eliminate this. Thin wall sections will tend to shrink less.
- Molding stress can make a part brittle and cause cracking. Lowering the injection velocity and increasing the mold temperature will improve this problem.
- Quenching will impart brittleness and cracks on many resins. Increase the mold temperature and lengthen the cooling time.
- Many hygroscopic resins will become brittle and crack on ejection if processed wet. Verify that all drying parameters are properly set for given resin.
- Increase the injection forward time, injection velocity and packing phase to ensure the part is full and weld lines are strong.
- Verify mold release and cleaners are compatible with the base resin.
- 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.

- Increase cooling time, decrease mold temperature and lower melt temperature so that the part will set quicker.
- Decrease injection/holding speeds and pressures to reduce the packing of the cavity.
- Increase draft on the tool's core, cavity and standing features to eliminate undercuts. Ensure that textured surfaces have enough draft.
- Increase packing of the part to maximize material input to the cavity and reduce shrinking to the core.
- Increase tool temperature on ejector half to induce more shrinkage to that side.
- Ask for additional lubricant on resin from material supplier or add mold coatings.

DELAMINATION

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

- Review this resin handling system to ensure no foreign materials are mixed with virgin or regrind. Switch to 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.
- Cross contamination with other resins or separation of alloys and blends can cause delamination. Ensure process is set properly for blends and alloys and look for contamination.
- Round all sharp corners or wall thickness changes. This will propagate laminar flow.
- Improve homogeneity of melt by increasing screw speed, back pressure or compression ratio of the screw. This will provide more shear heating and mixing melt.
- Adjust the mold temperature to the suggested range to the selected resin.
- Slow injection speed to reduce shearing and polymer separation.

DISCOLORATION

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

- Purge out injection barrel until resin runs clean.
- Lower melt temperature, screw speed and/or back pressure. Also, decrease injection velocity.
- Reduce cycle time, use small volume barrel. Use a reverse temperature profile.
- Verify feed throat temperatures and reduce feed zone temperature.
- Shot capacity is too small, move to large barrel volume to aid dispersion of additives and reduce severe shear heating.
- 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.

- Increase clamp pressure set point. Lower pack and hold pressures. Lower injection velocity.
- Reduce the shot size and/or increase the cushion.
- Lower melt temperature to prevent easy flow into vents and parting line.
- Increase drying of hygroscopic resins. Moisture can reduce molecular weight and increase flow.
- Begin packing with very low pressure to allow skin to set at points of flash and then ramp up the packing and holding pressures.
- Inspect and repair work or damaged regions of the parting line and vents.
- 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 or the tool.

- Increase melt temperature to aid flow, packing and improve reflection of tool surfaces.
- Locate gate to fill thickest section first and produce the smallest amount of weld lines.
- Raise mold temperature so skin does not set up as quickly and packing phase is effective in replicating mold surface.
- Ensure the cold slug wells are large enough to catch any frozen material from the nozzle. Add wells at the end of runners.
- Inject fast to push welds together while material is hottest.
- Clean vents and verify depth is correct for the given material.
- Increase packing pressure and time to push flow front together.

JETTING

Jetting or warming 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.

- Lower injection speeds or profile injection with slow initial speed, then ramp it up.
- Increase gate size to reduce nozzle effects.
- Move gate so that material impinges on a wall or boss immediately upon entering the cavity.
- Raise melt temperature to reduce viscosity and aid formation of flow front.
- Reduce the land length of the gate. Review material supplier recommendations for this dimension based on specific material.
- Raise nozzle temperature to ensure material is not solidifying.
- Incorporate "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 of leave surface defects on the molded parts.

- Reduce melt temperature of the resin, lower back pressure and/or screw speed.
- Reduce residence time of resin in the barrel by running a faster cycle or moving to a press with smaller shot capacity.
- Lower injection velocity, verify that gate sizes are proper and increase if necessary.
- Verify vent sizes are correct for the given resin. Increase the number of vents at more locations.
- Verify that mold surfaces are set at the material suppliers recommended range.
- 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 the cavities in the mold. Shorts can be caused by machine, mold or material.

- Adjust shot size and cushion size until part is properly filled.
- Raise injection velocity to push material into cavity before gate freezes off. Lower injection velocity if it is being trapped at the end of fill.
- Raise injection pressure.
- Increase melt temperature to ease injection. Raise barrel set points, screw speed and/or back pressure.
- Increase mold temperature to aid flow and prevent thick solid skin from setting.
- Ensure that 95% of cavity is filled by 1st stage injection and the remainder is filled and packed under 2nd stage (hold).
- Clean vents and ensure they are the proper size. Burning may also accompany this.
- Increase the backpressure so that adequate melt densification occurs.
- Verify clearances to prevent back flow.

SINK MARKS

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

- Increase shot size and/or decrease cushion to push more resin into the cavity. Ensure consistent cushion size.
- Increase the screw forward time under pressures to ensure gate freeze off is obtained.
- Slow the cooling/increase mold temperature so that the outré skin and the interior of the part shrink at the same rate.
- Decrease mold temperature on the side of the sink to thicken the frozen layer and retard the shrinkage around the sink mark.
- Lower the melt temperature to lower the amount of heat that needs to be pulled from the part. This will help reduce differential cooling and shrinkage.
- Ensure gating is from thick to thin to achieve the best filling and packing possible.
- Increase runner and gate sizes to get best pressure transfer to the part in the cavity of the mold.
- 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 of hang-up looks similar but is often accompanied by black specks or yellowness. Splay due to contamination is often accompanied by sever delamination.

- Check dryer's dew point, temperature and residence time. Check hoses for holes.
- Verify barrel set points and measure purge shot with a pyrometer. Lower is necessary.
- Lower injection velocity and screw speeds. Inspect nozzle, sprue, runners and gates for hang up points and sharpen corners. Ensure that screw design is not too aggressive.
- Turn decompression off.
- Verify venting is the proper depth for given resin.
- Inspect virgin and regrind for possible contaminations. Try running just virgin from another package.
- Increase mold surface temperature.

SURFACE FINISH

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

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

VOIDS / BUBBLES

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

- Increase injection, hold/pack pressures & time to ensure the cavity is completely filled through gate freeze-off, monitor part weight until it plateaus.
- Increase shot size and/or lower cushion to put more volume to cavity.
- Increase size of nozzle, sprue, runners and gates to allow for longer packing.
- Raise mold temperature to allow for slower cooling of skin. This will make material inside less likely to pull away from itself.
- Lower melt temperature to ensure part cures as fast as possible with minimal shrinkage.

HAITIAN ABSOLUTE