Southland Polymers
14030 Gannet Street, Santa Fe Springs, CA 90670 | Tel: 562-921-0444 | Fax: 562-921 0034

Injection Molding Processing Guide

Extruder Temperatures

Processing temperatures for a polyethylene resin are not specific but vary by injection molding machine. Extruder size, percent shot size, cycle time, number of mold cavities and screw back pressure all affect the set points for extruder barrel temperatures. Stated another way, molding temperatures depend on the residence time of the melt in the machine (i.e. the amount of time the polymer melt has been in the machine to absorb heat), how far the melt has to flow, and the amount of back pressure on the melt.

Large machines usually require lower temperature settings because of longer residence times for the polymer melt. Conversely, smaller machines and thinner parts (running a fast cycle) require higher temperatures to compensate for shorter residence times. A good rule-of-thumb for establishing extruder barrel temperatures is to start low, achieve mold fill, and then add 20F. Also, the extruder barrel temperature nearest the feed section may have to be set lower than the other barrel temperatures to prevent bridging (i.e. blocking) of the feed from the hopper.

For HDPE resins with a melt index value less than or equal to 10 g/10 min, a good starting point for extruder barrel temperatures is 450F. For HDPE resins with a melt index value between 10 and 30 g/10 min, a good starting point for extruder barrel temperatures is 425F. For HDPE resins with a melt index value greater than 30 g/10 min, a good starting point for extruder barrel temperatures is 400F.

Mold Temperature

It is to a molder's advantage to use the lowest mold temperature possible while avoiding condensation and inconsistent mold fill. The mold temperature at which condensation occurs depends on the relative humidity of the air. Increasing the mold temperature usually improves mold fill.

Injection Pressure

In molding polyethylene, it is usually to a molder's advantage to inject the melt into the mold as quickly as possible (using maximum injection pressure and fill rate) without flashing or overpacking the part. A good rule-of-thumb is to achieve 90% mold fill with injection high pressure and pack out the remaining 10% using the injection low pressure.

Back Pressure

Back pressure improves the mixing of the polymer melt but typically lengthens the overall cycle time due primarily to the resulting increase in the melt temperature. Therefore, back pressure should only be used for the mixing of

Southland Polymers
14030 Gannet Street, Santa Fe Springs, CA 90670 | Tel: 562-921-0444 | Fax: 562-921 0034

color concentrates or blends with dissimilar melt indexes. The minimum necessary to achieve mixing should be applied.

Mold Clamping Pressure

For polyethylene resins, approximately 2.5 tons of c lamp force is needed per square inch of projec5ed part area. If the clamp pressure is less than 2 tons/inch2, the injection pressure will overcome the clamping pressure resulting in the production of flashed parts.

Regrind

Polyethylene resins are typically characterized as being thermally stable, so !00% polyethylene regrind can be molded. Blends of regrind and base resin can be processed as long as the two components are from similar grades. A slight increase in back pressure may be required if the particle size/shape of the regrind is different from the base resin

Shrinkage Factors

The shrinkage factors listed below should be considered "ballpark" values since resins are not the only parameter that affect the shrinkage of a molded part. Other influencing factors are part geometry, processing conditions, and part temperature upon ejection from the mold.

Since these factors are "ballpark" values, the recommended procedure is to finish only one care and cavity using these values. Based on the measurements of parts form this single core/cavity combination, the shrinkage factor can be adjusted to fit the particular situation.

HDPE	0.020 – 0.022 inch/inch
LLDPE	0.017 – 0.019 inch/inch

Southland Polymers
14030 Gannet Street, Santa Fe Springs, CA 90670 | Tel: 562-921-0444 | Fax: 562-921 0034

Injection Molding Trouble Shooting Guide

Defect	Possible Cause(s)	Suggested Remedies
Short Shots	Insufficient feed	Increase feed / shot size
	Melt freezes too soon	Fill mold before resin solidifies
	Improper injection conditions	Increase injection time Increase injection pressure Use nozzle with larger opening Increase sprue, runner or gate size Decrease gate land length Add more gas vents Change gate location
	Melt freezes too soon	Increase barrel temperatures Increase mold temperature Increase injection rate Use resin with higher melt index
Sink Marks	Insufficient packing	Increase injection pressure Increase injection time Increase feed Increase sprue, runner or gate size Decrease gate land length Change gate location
	Thick sections too hot	Decrease barrel temperatures Increase mold close time Decrease injection rate Use resin with higher melt index
Flash	Too much feed	Decrease feed / shot size
	Melt viscosity too high	Increase barrel temperature Decrease injection pressure Decrease injection velocity Increase clamp pressure Use resin with higher melt index
	Melt viscosity too low	Decrease barrel temperature Use resin with lower melt index and / or higher density

Southland Polymers
14030 Gannet Street, Santa Fe Springs, CA 90670 | Tel: 562-921-0444 | Fax: 562-921 0034

Defect	Possible Cause(s)	Suggested Remedies
Short Shot & Flash	Cold / viscous melt	Increase barrel temperatures Increase screw back pressure Increase mold temperature Decrease injection pressure Decrease injection rate Increase clamp pressure Increase sprue, runner or gate size Use resin with higher melt index
Voids / air Bubbles	Insufficient mold venting	Clean vents Add more gas vents Increase injection time Decrease injection rate
	Melt degassing	Increase screw back pressure Increase feed Increase extruder feed zone temperature
Weak Weld Lines Or Flow Marks	Cold / viscous material	Increase injection rate Increase injection pressure Increase mold temperature Increase barrel temperatures Use resin with higher melt index Decrease gate land length Increase sprue, runner or gate size Use nozzle with larger opening Change gate location
	Insufficient mold venting	Clean / add vents
Brittle Parts	Excessive part packing	Decrease injection pressure Decrease injection time Decrease barrel temperatures Increase mold temperature Decrease injection rate Decrease feed Decrease gate land length
	Resin selection	Use resin with lower melt index and density

Southland Polymers
14030 Gannet Street, Santa Fe Springs, CA 90670 | Tel: 562-921-0444 | Fax: 562-921 0034

Defect	Possible Cause(s)	Suggested Remedies
Warpage	Part too hot	Increase mold close time Decrease barrel temperatures
	Overpacking part	Adjust feed Decrease injection pressure
	Resin selection	Use resin with narrower MWD
Poor Mold Release	Overpacking part	Decrease injection pressure Decrease injection time Increase injection rate Decrease feed
	Excessive shrinkage	Decrease barrel temperatures Increase mold temperature
	Equipment Problem	Decrease gate land length Reseat machine nozzle Sandblast or vapor hone mold
	Resin selection	Use resin containing slip
Excessive Shrinkage	Part too hot	Decrease barrel temperatures Decrease mold temperature Increase mold close time Increase injection time Decrease injection rate Increase injection pressure
Poor Color Dispersion	Poor mixing	Increase screw back pressure Increase injection rate
Sprue Sticking	Overpacking part	Decrease injection pressure Decrease injection time
	Equipment problem	Reseat nozzle Resurface sprue Increase taper of screw
Burn Marks	Inadequate venting of gasses	Clean vents Decrease injection rate Decrease barrel temperatures Decrease injection pressure Add more gas vents