Injection Molding Polypropylene

1. *Injection Machine Requirements*

Rated Capacity : An estimate should be made of the shot size "ounces" required for the particular molded part. This estimate should be compared with the machine's rated capacity. Polypropylene shot sizes have a maximum limit of approximately 75% of the machine' rated capacity in polystyrene.

Screw Type : A typical screw used for polypropylene is the single-stage, general purpose type with a length-to-diameter (L/D) ratio of 16:1 to 24:1 and a compression ratio between 2.5:1 and 3.0:1. A two stage screw with a vented barrel is not required. Barrier and mixing type screws work best for colorant dispersion and fast cycling. Generally, reverse temperature profile heat setting are usd with barrier screw designs.

Screw Decompression: To eliminate drool from the nozzle, use decompression to help set the slip ring style NRV. 0.1 to 0.3" of decompression is suggested.

Clamping Force: The clamping force required for polypropylene is 2 tons/in.^{2} (2.8 kg/mm^2) of projected area for a wall thickness greater than or equal to 0.090 in. (2.3 mm). Smaller wall thicknesses require slightly more clamp tonnage; therefore, an overall average clamp tonnage of 2.5 ton/in.^{2} (3.5 kg/mm^2) is commonly recommended for parts with wall thicknesses less than 0.090 in. (2.3 mm).

2. *Startup Molding Conditions*

Drying: Polypropylene(unfilled) may contain minimal surface moisture, but is not considered hygroscopic. Although drying is not usually recommended, 2 hours at 140°F will dry excessive surface moisture.

Melt Temperatures : Factors influencing the choice of injection melt temperatures to be used are: Part geometry, runner and gate sizes, barrel size, shot weight, machine capacity ratio and type of resin. The following melt temperatures are good starting points:

Flow Characteristics	*Melt Temperature Range, °F (°C)*
Low flow	475-525 (246-274)
Medium flow	445-475 (228-274)
High flow	400-445 (200-228)
Very high flow	385-420 (196-215)

Use of hand held melt pyrometers is strongly recommended because barrel temperatures do not always reflect the true temperature of the melt. Most barrel thermocouples measure steel "barrel" temperatures instead of the melt temperature. The barrel temperature profile should be set so that there is a progressive increase in temperature from the hopper to the nozzle. **Excessive barrel and melt temperature and long residence times within the barrel can cause the natural color to turn yellow which will further cause finished part color shifts.**

Mold Temperature: The startup temperature for the mold is normally room temperature. For good surface appearance of the part, mold temperatures of the front(A) plate should be between 70 and 120°F (21 and 49°C) and between 60 and 100°F for the rear (B) plate. However, the most suitable mold temperature will be determined by the individual mold, part dimension and molding cycle.

Injection Booster pressure: The booster pressure should be the maximum that can be achieved without flashing, and is best determined by the melt temperature employed. Ideally, the booster pressure should be approximately 60% of the machine maximum, 2,000 psi (14 MPa). This translates into a hydraulic pressure reading of 1,200 psi (8.3 MPa) or a plastic pressure of 12,000 psi (83 MPa). The melt temperature should not be decreased to the point where excessively high injection pressures must be used.

Injection Hold (Low) Pressure: The injection hold pressure is frequently set lower than the booster pressure. This should provide complete packingof the mold and allow the gate to freeze off.

Back Pressure: Screw back pressure is commonly employed to improve colro dispersion and melt uniformity. The effect is one of increasing the amount of work going into the mix. However, Screw recovery rates decrease with increased back pressure. Typical back pressures used in injection molding polypropylene are between 50 and 300 psi (0.3 and 2.1 MPa).

3. ***Optimizing Mold Cycles for Maximum Output***: The minimum cycle that will maximize productive output can be determined by studying each time component in the following order:

Injection Hold (2nd Stage) Time: To reduce cycle time and thereby maximize output per unit time, the injection hold time should be decreased in 1 sec. Increments until a minimum time is reached (as long as necessary to freeze off the gate).

Injection Booster (1st Stage) Time: Booster time should be held to a necessary minimum. Although a fast fill is often recommended, it has been observed in some cases that slower fill has correct many molded part defects and may be a preferable technique.

Cure Time: The cure, or cooling, time should be reduced in 1 sec. Increments. The limitations to reduction of this time component are the appearance and dimensions of the mold part. The minimum cure time can be determined by a close inspection of the parts produced after each incremental change.

Part Removal Time: This time component is minimized by an incremental decrease of the mold open and close times. It can be further minimized by an increase in ejection speed, if possible.
A careful and methodical approach to cycle time reduction following the above steps will help the molder minimize production costs.

4. ***Use of Regrind***: Excess molded material (runners, sprue, gates, etc.) and part rejects from similar resin grades can be reground and processed in a blend with virgin material. Regrind polypropylene should be considered, from a properties standpoint, as being different from the virgin material. Depending on the severity of the molding conditions, the reground material can have a higher melt flow rate, possible discoloration (yellowness) and slightly lower physical properties such as stiffness, impact strength, tensile strength and elongation. As a starting point, it is recommended that approximately 20% regrind be used. However, the molder must experiment to arrive at the optimum blend ration that can be molded with acceptable part performance and appearance.

Troubleshooting Common Injection Molding Problems

1. Short Shots (Parts Not Filling)	***Possible Causes*** Insufficient injection pressure Insufficient Injection time Insufficient feed Unbalanced multiple cavity mold Foreign material clogging nozzle and/or gates Too low stock temperature Too small runners, gates, or vents Too low mold temperature Undersized cylinder heating capacity
2. Mold Flash	***Possible Causes*** Too high injection pressure Foreign material on mold surface Low clamping pressure Too high stock temperature Bad mold parting line Excessive injection velocity
3. Excessive Shrinkage	***Possible Causes*** Too short cure time Too low pack pressure Too high mold or stock temperature Too small runners or gates Poor part design, varying wall thickness
4. Warpage	***Possible Causes*** Too hot ejected part Improperly balances core & cavity temperature Inadequate or poor location of knockout mechanism Overpacking in gate area because of high injection pressure
5. Brittleness	***Possible Causes*** Degraded material from cylinder Contamination Improper design; inadequate radii at corner, notch, or thread Use of improper color concentrates (made from another resin) Voids Too low stock and mold temperature

NETWORK POLYMERS, INC.

Troubleshooting Common Injection *Molding Problems (continued)*

6. Flowmarks, Weldlines, Knitlines	***Possible Causes*** Excess mold lubricant Too fast mold fill, inadequate venting Too low injection pressure and packing time Improper gate location or design
7. Low Gloss, Rough Surface	Too low stock temperature Too low or non-uniform mold temperature Scratched or dirty mold surface Poor pigment dispersion, Inadequate venting Excessive moisture on pellets
8. Sinkmarks	***Possible Causes*** Insufficient injection pressure Insufficient dwell time Poor part design, non-uniform walls &/or excess wall thickness Too high mold temperature Improperly located/designed gate
9. Erratic Quality	***Possible Causes*** No cushion or back pressure Non-uniform feed temperature Non-uniform cycle Erratic equipment performance Unbalanced multiple cavity layout and runner system Undersized cylinder; insufficient volumetric and heat capacity
10. Voids	***Possible Causes*** Failure to fill mold completely (see Short Shot) Poor venting of mold, particularly around projections Improper location of gate Too rapid fill rate Excessive moisture on pellets
11. Sticking in Mold	***Possible Causes*** Overpacking; too high injection pressure Polished surface on core Insufficient knockout action Surface irregularities in the mold Insufficient core and wall tapers