INVISTA™
POLYPROPYLENE

Injection Molding Polypropylene

The information set forth herein is being provided for informational purposes only. The information is provided on a without prejudice basis and should not be viewed as giving technical advice, instruction, or otherwise. The information is furnished free of charge and is based on INVISTA Polypropylene's knowledge and understanding. INVISTA Polypropylene *makes no representation or warranty as to the completeness or accuracy of the information contained herein*. It is intended for use by persons having technical skill, at their own discretion and risk, who will make their own determination as to its suitability for their purposes prior to use. As with any material, evaluation of any compound under end-use conditions prior to specification is essential.

INVISTA Polypropylene
118 Huntsman Way – Longview, Texas 75602 Tel: 903-239-5200

Polypropylene introduction

Polypropylene is a versatile thermoplastic resin available in a wide range of formulations from general purpose homopolymer, random copolymer and impact copolymer grades to highly specialized resins for engineering applications. Various grades of polypropylene are available to meet the needs of various applications using the injection molding process. The wide range of physical properties and relative ease of processing makes polypropylene an extremely attractive material, capable of competing with more expensive resins in a number of demanding applications.

Material selection

Because of the diversity of polypropylene types and formulations resulting in different mechanical and optical properties balance, it is recommended that you consult the INVISTA technical department to review the needs of your application when selecting one of our polypropylenes. Things to consider in material selection are:

- Is your part colored?
  - Multiple colors?
- Will you sterilize your parts and if so, which process will be used?
- Post mold assembly also needs to be considered as another example.
  - Will you be printing?
- What are your regulatory requirements?
  - Food contact
  - UL requirements

Molding Machine Considerations

Polypropylene can be successfully molded with either plunger or reciprocating screw injection molding machines, including single and two-stage machines. A reciprocating screw machine is generally preferred for applications requiring melt homogeneity.

Injection machine requirements include 20,000 psi injection high pressure and separate injection speed control. Injection speed and hold pressure profiling can also be of benefit. Using the lowest pressure that fills the mold can extend mold life. This will also reduce flashing.

Clamping forces between 1½ and 5 Tons/square inch have been used. Low viscosity (high melt flow rate) resins generally require lower tonnages than high viscosity resins. To increase and maximize processing window, one should design towards a marginal higher tonnage than needed. While a general-purpose screw can be used, many applications benefit from screw engineering to promote rapid melting and improved dispersion. Shot size should be 50 to 75% of barrel capacity.

Auxiliary Machines

Central and press-side auxiliary machines can be used for drying, mold temperature control and sprue and runner grinding. Selection will rely on individual plant engineering.

Drying
Polypropylene resins typically do not require drying before processing. Touch-up drying may be required to reduce and eliminate splay from condensed moisture, as when resin is brought into an area with high humidity from cold transport or storage. In very critical applications, it may be necessary to pre-heat the material so it is consistent. Pre-heating can reduce shear heating in the screw and allow for better melt temperature stability.

Mold Temperature Control
Heaters and or chillers will achieve mold temperature control. Low mold temperatures can give faster cycles, with the concern for molded in stresses. Higher mold temperatures will give more complete replication of mold surface features. Water or water/glycol mixtures are sufficient for most polypropylene molding requirements.

Grinders
Most conventional grinders are suitable for regrinding sprues, runners, and rejected parts. Screen diameters should be ~¼". Each application should be evaluated to determine the maximum acceptable level of regrind. The regrind should be kept clean, uncontaminated, and be well-blended with virgin resin before molding.

Mold Design Highlights

Mold Materials
Molds for polypropylene have been successfully made from many materials. Production molds are typically made from hardened steel, using pre-hardened bases and either stainless or higher hardness tool steel for the cavity and core inserts. Specialty inserts have been made from high conductivity materials such as copper alloys and/or self-venting porous materials. Prototype molds have been made from diverse materials such as aluminum, nickel coated epoxy, and cast zinc alloys.

Mold Design
The **nozzle and sprue bushing** create the transition from injection molding machine to the mold. The nozzle and sprue need to be matched for both spherical radius (nozzle locating) and nozzle exit to sprue entrance diameter, for both cold sprue and hot sprue bushing applications. For cold sprue and runners, the nozzle will have an exit orifice typically ¼" diameter and the entrance of the sprue bushing will be 1/32" diameter larger. These dimensions will vary with specific parts and are intended to allow for the inevitable mis-match. The sprue and bushing diameters will probably be larger for hot sprue & runner applications. Matching these elements will aid part ejection and color transitions. Whether hot or cold runner systems are used, the runner should be as short and direct as possible. Runner and cavities should be balanced for multi-cavity molds. **Gate location** should be selected to minimize sinks, voids and weld lines. General practice is to gate the part into its thickest section. Many **gate designs** have been successfully used, among them - pinpoint, tunnel, cashew, rectangular, diaphragm or flash, and full round. Whatever gate design is selected, a smooth, tapered transition from runner to gate and short land length (~0.020") is preferred. Gate diameters range from 50% to 75% of the part thickness at the gate. To prevent sinking, a lower ratio of gate to part wall thickness is desirable. **Cavity and runner venting** are essential for smooth, rapid filling and easy molding. Vents can be located on the parting line, along ejector pins, or with inserts made of porous mold materials as needed. Vent sizes can start at 0.0005" thick x 0.020" land x width to suit. In general, vents will increase in depth away from the cavity edge.

INVISTA Polypropylene Mold Shrinkage

Allowance for mold shrinkage must be made when designing a mold for a part that requires close dimensional tolerances. Mold shrinkage of thermoplastic materials is very complex because it is affected by so many factors.

For polypropylene, a major factor is cooling rate. Generally, higher shrinkage's result from slower cooling rates, which is why thicker parts shrink more than thinner parts. It simply takes longer to remove heat from thicker parts.

Other factors affecting part shrinkage are melt temperature (which affects cooling rate), packing of the part, and the addition of fillers to the polypropylene.

Polypropylene will continue to shrink and crystallize for several days or weeks after molding. Studies of shrinkage versus time show that most of the shrinkage takes place within the first 24 hours after molding. A small amount of additional shrinkage will occur in the next 24 hours, followed by incremental amounts (that are difficult to measure) for two to four weeks. Exposure of molded parts to elevated temperature or humidity can also cause additional part shrinkage.

Mold shrinkage for polypropylene can vary from about 0.010 mm/mm (0.010 in./in.) to about 0.030 mm/mm (0.030 in./in.) or 1-3% of dimension, depending on part thickness, formulation, and processing conditions.

For parts requiring close tolerances, a prototype mold should be made so that some actual shrinkage measurements can be obtained before the production mold is built.

Molding Pointers

Back Pressure can range from 50 to 250 psi with machine capability. Increased back pressure will lengthen screw recovery time and increase mixing of additives such as colorants.

Fast **Injection Speeds** will generally give better results than slow speeds. This can vary from one application to the next. Increased packing will generally give stronger and stiffer parts than under-packed. Caution is needed, as over- packed parts can be brittle.

Injection Pressures and Fill Rate should be as high as possible. Transfer to hold should be set at 95 to 97% cavity fill.

Hold Pressure should be 50 to 75% of Injection Pressure. Set **Hold Time to finish at Gate Freeze** to ensure packed parts.

Melt Temperatures: this will vary depending on polypropylene Melt Flow Rate, tool cavitation and part thickness. The table below will give you good starting points. It is recommended that you actually measure your melt temperature with a pyrometer when you set-up a new mold with a given polypropylene.

Melt Flow Rate (g/10 min)	**Melt Temperature Range, ^{0}F (^{0}C)**
Low, 2 to 5	475-500 (246-260)
Medium, 5 to 20	440-475 (226-246)
High, 20-35	400-445 (200-228)
Extra high, greater than 35	370-420 (185-215)

Mold Temperature:
The proper mold temperature will vary with individual mold, part dimension and thickness as well as molding cycle. Typical temperature will range between 60 and 120 ^{0}F (15 to 49 ^{0}C).

Trouble Shooting Guide

Problem	Causes	Solution
Black Specks*	Polymer degradation	• Use proper process temperature • Lower barrel set-up temperatures when experiencing machine down time. If excessive machine down time is unavoidable, use air shots to purge barrel
	Regrind	• Use clean regrind(potential contamination point) • Insure regrind has minimal amount of fines
**Consult your vendor for material specification*	Improper sizing of screw and barrel	• Optimize residence time in the barrel to match shot size. Typically 2.5 shots in the barrel is recommended
Brittleness	Improper material selection	Consult your polymer supplier
	Residual part stresses	Insure proper part design Avoid over-packaging at the gate
	Polymer degradation	Refer above
Bubbles (voids or trapped gas)	Voids	Non-uniform wall thickness Internal part pressure is greater than external pressure Increase pack pressure and hold time
	Trapped gas	Verify material is dry
		Tool vents should be open and clean with proper location
		Use fastest injection speed possible without causing other issues
		Reduce feed zone temperature to allow gas to back flow and escape
Discoloration *# Consult with material vendor for the appropriate material selection*	Yellowing	Melt temperature too high
		Residence time too long
		Improper material selection #
		Using too much regrind
	Dark streaks	Refer to black speck section
		Verify absence of contamination
		Miss alignment of nozzle and sprue bushing

Dimension control	Parts too small	Increase hold pressure
		Improper material selection
		Tool design
	Parts too large	Reduce hold time and pressure
		Reduce melt temperature
		Tool design
		Improper material selection
	Warpage	Internal part stress
		insufficient balanced cooling
		Unbalanced part ejection
		Differential wall thickness
		Over packing
		Cycle too short
	Sinks	Mold design
		Insufficient packing
		Gate design too small
		Reduce fill rate
		Backside features on part too thick
Ejection problems (sticking)	Sticking in the cavity	Reduce packing
		Insufficient cooling
		Insure differential cooling between the core and the cavity (more cooling on cores)
		Increase polish on cavity
		Grade selection
	Sticking on cores	Reduce packing pressure
		Reduce hold time and/or pressure
		Improve the ejection system
		Consider adding mold release (can be added to the polymer itself by the polymer vendor) or at machine side
Knit lines	Weak knit lines	Increase melt temperature
		Mold temperature too cold
		Increase injection speed
		Avoid nucleated products if possible
		Inadequate venting
		Mold design: position gate placement to create knit lines in non-critical areas

Unmelts	Improper melt temperature	Increase melt temperature; use back pressure increase at first to increase melt temperature in small increment
		Preheat the material
	Improper screw/barrel capacity for part size	Re-size injection molding equipment
	Contamination	Verify the material is not contaminated
Gate stringing	Manifold design	Consult your equipment vendor
	Hot runner temperature	Lower temperature
	Hot runner nozzle tip	Reduce the temperature to near freeze off
	Material selection	Narrower molecular weight polymer selection
Excessive static	Static generated during mold filling	Lower injection speed
		Raise melt temperature
		Raise mold temperature
		Increase cycle time
	De-nesting problems	Part design
		Material selection (adding antistat)
		Secondary static dissipating equipment