INTRODUCTION

The intent of this technical brochure is to provide molders with a general insight into the characteristics of polypropylene and aspects of injection molding processing for optimum results. Prospective molders should consult with a CPC technical representative for questions which can't be answered with this free guide.

MOLDING PARTS WITH POLYPROPYLENE

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's rated capacity in polystyrene

Screw Type

A typical screw used for polypropylene is the single-stage, general- purpose type with a length: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.

Clamping Force

The clamping force required for polypropylene is 2 tons/in.2 (2.8 kg/mm2) of projected area for a wall thickness greater than or equal to 0.090 in. (2.29 mm). Smaller wall thicknesses require slightly more clamp tonnage; therefore, an overall average clamp tonnage of 2.5 tons/in.2 (3.5 kg/mm2) is commonly recommended for parts with wall thicknesses less than 0.090 in. (2.29 mm).

Startup Molding Conditions

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 flows	475-525 (246-274)
Medium flows	445-475 (228-274)
High flows	400-445 (200-228)
Extra-high flows	385-420 (196-2 15)

Following is a simplified rating of the available melt-flow ranges into flow characteristics:

Melt-Flow Range, g/ 10 mm	**Flow Characteristics**
Below 4	Low flows
4 to 10	Medium flows
10to 20	High flows
Above 20	Extra-high flows

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 melt temperature.

The barrel temperature profile should be set so that there is a progressive increase in temperature from the hopper to the nozzle.

Mold Temperature

The startup temperature for the mold is normally room temperature. For good surface appearance of the part, mold temperatures between 70 and 120°F (21 and 49°C) are generally used. 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 melttemperature employed. Ideally, the booster pressure should be approximately 60% of the machine maximum, 2,000 psi (13.8 MPa). This translates into a hydraulic pressure gauge reading of 1,200 psi (8.3 MPa) or a plastic pressure of 12,000 psi (82.8 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 packing of the mold and allow the gate to freeze off.

Back Pressure

Screw back pressure is commonly employed to improve color 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).

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 for the injection molding of Pro-fax polypropylene, it has been observed in some cases that a slower fill has corrected 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 imitations to reduction of this time component are the appearance and dimensions of the molded 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 steps just listed will help the molder minimize production costs.

Use of Regrind

Excess molded material (runners, sprue, gates, etc.) and part reects from similar resin grades can be reground and processed in a blend with virgin material.

Regrind polypropylene should be considered, from a property 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 ratio that can be molded with acceptable port performance and appearance.