

Molded Part and Tool Design

Cooling. Mold cooling is absolutely critical to reduce cycle times, prevent sticking, and aid in part ejection. Cooling is especially critical in cores, corners, and tall-standing details and near gates with hot drops. Where possible, use thermally efficient alloys such as Ampco™ 940 or Moldmax® for cores and other hard-to-cool areas of the mold. Minimize looping of cooling lines and design all cooling circuits for adequate turbulent flow.

Cold Sprue Design and Cooling. Polyester-based polymers tend to stick to hot steel. To prevent sprue sticking:

1. Use a slight interference fit of 0.005 mm (0.0002 in.) between the sprue bushing and mold plate to ensure good heat transfer. Maintain line-to-line fit along the entire length of the sprue bushing shaft.
2. Use thermally efficient alloy bushings, such as those made by Performance Alloys and Services, Inc.
3. Keep sprues shorter than 80 mm (3 in.) with an included draft angle of at least 2°.
4. Polish the sprue in the draw direction.

Cold Runner Design. When designing cold runner systems, use the same guidelines that apply to most engineering resins. This includes designing for smooth and balanced flow, generously radiused transitions, cold slug wells, runner venting, and full- or half-round cross sections.

Gate Design. Gates should be large enough with minimal sharp transitions to minimize gate blush and prevent premature gate freeze. Edge gates and tab gates should be 50% to 80% of the nominal wall thickness. Typical tunnel gates are at least 1.6 mm (0.065 in.) diameter, while typical pinpoint gates in three-plate molds are 0.75 to 1.1 mm (0.030 to 0.045 in.) to minimize the vestige. Where part size and cooling allow, pinpoint gating may be from inside the part or into a recess to hide the vestige.

Hot Runners/Hot Drops/Valve Gates. PET polymers work well in hot runner/hot drop and/or valve-gated molds that are properly designed for amorphous polymers. Many vendors have designs specifically for polyester-type polymers. In general, the types that work well have heaters external to the melt stream with no spots for polymer hold-up. Also, drops with integral cooling around the gate give more consistent temperature control and vestige formation.

Types of hot drops **not recommended** because of the potential for degradation are those that have heaters *internal* to the melt stream but no heaters or insulation *external* to the melt stream. Some polyester materials like PET tend to crystallize and whiten at the gates in hot runner systems. This may require gate placement into noncritical areas or into a post or tab that can be hidden or removed.

Venting. Typical vent depth is 0.012 to 0.025 mm (0.0005 to 0.001 in.) for small parts or vents close to gates and 0.025 to 0.038 mm (0.001 to 0.0015 in.) for larger parts. Land length is typically 3 to 6 mm (0.125 to 0.250 in.). Polished vents are suggested to minimize build-up when molding formulas containing mold release.

Mold Polish/Coatings. Mold surfaces should be polished to a level adequate for ejection (SPI 5 or lower) and to a smoother polish only if required for aesthetics (SPI 3 or lower).

On drafted walls, do not use a polish that is too smooth as this may create a vacuum during ejection.

Coatings may be used where cooling is not adequate to prevent sticking. We suggest nickel/phosphorous/PTFE or tungsten disulfide coatings

Draft Angles and Textured Surfaces. In polished molds, 1° draft per side is suggested to aid ejection. However, 0.5° draft can be used to obtain reasonable dimensions in ribs, bosses, or other design features. Avoid using zero draft in any part of the mold. Texturing requires increasing the draft angle by 1° to 1.5° for every 0.025 mm (0.001 in.) of texture depth.

Ribs and Radii. Ribs and walls should meet with a minimum radius of 0.5 mm (0.02 in.) or 0.2 to 0.5 times the wall thickness, whichever is greater. This will minimize stress concentration in molded parts and disturbance of flow during filling.

Undercuts. Must be rounded and well filleted to allow proper ejection.

Molding Machine

Shot Capacity/Clamping Tonnage. Actual shot size for molded parts, including runners, should be 40% to 70% of shot capacity to provide adequate plasticating time without excessive hold-up time. Clamping force is typically 400 to 700 kg/cm² (3 to 5 tons/in.²) of projected area of the part and runner.

Injection Profiling. The molding machine should be capable of profiling injection speed. Profiling is generally necessary to optimise the filling stage of the injection molding process.

Screw and Barrel Design. PET generally cause little wear to the screw and barrel, and no corrosion is expected.

1. General-purpose screws with compression ratios of 2.8:1 to 3.2:1
2. Length-to-diameter ratios of 18:1 to 20:1 have been used successfully. However, barrier-type screws may be beneficial when molding PET polymers on higher output machines.
3. Vented barrels are not recommended.
4. Ring-type nonreturn valves are preferred over ball-type valves to minimize hold-up time and degradation of polymer.

Nozzles. Reverse-taper nozzles are typically used for PET polymers

Processing

Consult your Marplex Australia Pty Ltd data sheet for specific drying conditions, melt temperature range, and mold temperature range for specific Marplex polymer. Data sheets are available online at www.marplex.com.au

Drying Conditions. All Marplex PET grades require predrying to minimize polymer degradation and/or to minimize bubbles or splay. Use desiccant-type dryers with a dewpoint capability of -30°C (-20°F) and carefully maintain the dryer according to the manufacturers guidelines.

Melt and Mold Temperatures. Actual melt temperatures for ASTAPET™ PET grades typically range from 250° to 280°C

Use the minimum temperature necessary to fill the part without excessive speed or pressure. Typical mold temperatures for ASTAPET™ PET80 and PET70 range from 15° to 25°



Marplex Australia Pty Ltd

ABN 78 004 691 614 ACN 004 691 614

221 Browns Road, Noble Park
Victoria 3174 Australia
PO Box 42, Sandown Village
Victoria 3171 Australia
Phone: +61 3 8710 1400
Fax: +61 3 9795 2111
Web: www.marplex.com.au

Injection Speed. Generally, use slower initial injection speed than for other plastics to minimize gate blush and/or splay. Inject at 10% to 20% of available speed for the first 10% of cavity fill, then ramp up to 40% to 80% speed to complete the shot.

Cushion. Keep cushion size at the absolute minimum to prevent degradation from excessive hold-up time. Typical cushion is 3 to 13 mm (0.12 to 0.50 in.).

Back Pressure and Decompression. Typical backpressure is 0.7 to 1.0 MPa, although higher pressures up to 2 MPa may be needed to eliminate air entrapment and improve melt uniformity. In general, little or no decompression is used.

Purging and Shutdown. The most effective purge materials are those similar to the polymer being run. **Do not use polyethylene or polypropylene purge materials.**

Always leave the screw forward after shutdown to ease restarting the machine. When changing from ASTAPET™ PET to another polymer, purge with Polycarbonate, acrylic, styrene, or commercial purge compound. Then run the screw dry and turn off the power.

Regrind. Clean, dried regrind can be used at blend rates up to 100%, although 10% to 30% is more typical. Keep blend rates uniform to minimize process variation. Since regrind has somewhat lower physical properties than virgin polymer, be sure that parts molded with regrind meet the end-use requirements.

Drying uncrystallized regrind blended with crystalline pellets may require special conditions and equipment. Always consult Marplex Australia Pty Ltd Technical Service