EASTMAN

Keys to **successful molding**

EASTAR™ copolyesters

Eastman CRISTAL™ EN copolyester

DuraStar™ polymers

Keys to **successful molding**

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 Ampcoloy™ 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:

- Use upper and lower cooling circuits around the sprue.
- 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.
- Use thermally efficient alloy bushings, such as those made by Performance Alloys and Services, Inc.
- Keep sprues shorter than 80 mm (3 in.) with an included draft angle of at least 2°.
- 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%–80% of the nominal wall thickness. Typical tunnel gates are at least 1.6 mm (0.065 in.) in diameter, while typical pinpoint gates in three-plate molds are 0.75–1.1 mm (0.030–0.045 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. Eastar™ copolyesters, Eastman Cristal™ EN copolyester, and DuraStar™ 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 holdup. 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 DuraStar or 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 0.012–0.025 mm (0.0005–0.001 in.) for small parts or vents close to gates and 0.025–0.038 mm (0.001–0.0015 in.) for larger parts. Land length is typically 3–6 mm (0.125–0.250 in.). Polished vents are suggested to minimize buildup when molding formulas contain 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/phosphorus/PTFE or tungsten disulfide coatings such as Poly-Ond™, Dicronite DL-5™, WS2™, or Nicklon™. Another to consider is Dylyn™/DLC which offers good wear resistance as well as low coefficient of friction, which improves release of molded parts on tiny cores or bosses where it is difficult to provide cooling water.

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°–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. Stripped undercuts, such as "rice grains," snap rings, or threads, are allowable up to 2%–3% of the part diameter in relatively thin-walled parts. 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%–70% of shot capacity to provide adequate plasticating time without excessive holdup time. Clamping force is typically 400–700 kg/cm² (3–5 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 optimize the filling stage of the injection molding process.

Screw and barrel design. Eastman polymers generally cause little wear to the screw and barrel, and no corrosion is expected. General-purpose screws with compression ratios of 2.8:1 to 3.2:1 and length-to-diameter ratios of 18:1 to 20:1 have been used successfully. However, customized screw designs have been used for specific needs such as improved color mixing and high-volume output. Many screw vendors offer specialized designs to improve performance for these types of needs. Vented barrels are not recommended. Three-piece, full-flow, ring-type nonreturn valves are preferred over ball-type valves to minimize holdup time and degradation of polymer.

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

Processing

Consult your Eastman representative or the formula datasheet for specific drying conditions, melt temperature range, and mold temperature range for specific Eastman polymers. Datasheets are available online at www.eastman.com.

Drying conditions. All Eastar and Cristal EN copolyesters, and DuraStar polymers require predrying to minimize both polymer degradation and/or bubbles or splay. Use desiccant-type dryers with a dew point capability of –30°C (–20°F) and carefully maintain the dryer according to the vendor's guidelines.

Melt and mold temperatures. Actual melt temperatures for Eastar, Cristal EN, and DuraStar typically range from 250° to 280°C (480° to 540°F). Use the minimum temperature necessary to fill the part without excessive speed or pressure. Typical mold temperatures for Eastar DN, EN, and GN, and Cristal EN copolyesters range from 15° to 25°C (60° to 80°F). Eastar AN and BR copolyesters and DuraStar polymers may benefit from somewhat higher temperatures, up to 40°–50°C (100°–120°F), depending on aesthetic and filling requirements.

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

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

Back pressure and decompression. Typical back pressure is 1.0–1.3 MPa (150–200 psi), although higher pressures up to 2 MPa (300 psi) 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 Eastman polymers 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%–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. Consult your Eastman representative for proper drying requirements.***

EASTMAN

The results of insight™

Eastman Corporate Headquarters
P.O. Box 431
Kingsport, TN 37662-5280 U.S.A.

U.S.A. and Canada, 800-EASTMAN (800-327-8626)
Other Locations, +(1) 423-229-2000

www.eastman.com/locations