Eastar™ copolyesters

Some dos and don’ts for injection molding

Eastar™ copolyesters are Eastman thermoplastic materials that have potential in a variety of injection molding applications because of their clarity, toughness, and outstanding price/performance ratio. Some dos and don'ts that should be helpful in processing this material follow.

Drying

DO

- Keep drying system clean to prevent contamination of Eastar copolyesters.
- Dry thoroughly with dehumidified desiccant drying system to prevent:
  - Bubbles or appearance problems on finished product
  - Degradation due to hydrolysis, which causes loss of toughness
- Use process drying air with a dew point of –30°C (–20°F) or less.
- Use air temperatures of 65°–70°C (150°–160°F). Air temperatures below 65°C (150°F) will substantially increase drying time. Air temperatures above 70°C (160°F) may cause the pellets to stick together.
- Dry the material for a minimum of 6 hours.
- Check dehumidifying system with a dew point tester.
- Keep desiccant bed clean.
- Check heater circuits and elements.
- Control temperature precisely.
- Use air-drying volume of 1.0 cfm/lb (0.06 [m^3/min]/kg) of material processed per hour.

DON'T

- Attempt to decrease drying time by raising temperature above 70°C (160°F). This can cause pellet sticking and bridging.
- Remove dried material from the drying system until just prior to processing. The moisture level increases rapidly when Eastar is exposed to ambient conditions. Even in the hopper, the material will begin to absorb moisture. Therefore, it is good practice to keep the material in the hopper as dry as possible.

Injection molding

DO

- Keep the material free from contamination to produce tough, clear parts.
- Purge the machine thoroughly, preferably with Eastar or with undried acrylics, polystyrene, commercial purging compounds, or the polymer to follow.
- Use standard molding equipment.
- Use conventional mold-design engineering to include full-round, large runners; large gates; and generously radiused corners.
- Use slow screw speeds of 30–60 rpm.
- Use slow injection speed to reduce streaking and gate splay. If injection speed is programmable, a slow/moderate/slow profile may be used to produce the best appearance and a reasonable fill time.
- Use lowest possible melt temperature to achieve satisfactory appearance and fill. Normally, the suggested operating range is 250°–270°C (480°–520°F), depending on dwell time of the material in the cylinder.
- Plan shot size to be at least 50% and preferably 75%–80% of machine capacity.
- Expect properties to be dependent on the interplay of temperature, time, and moisture.
- Allow for shrinkage of 0.003–0.005 mm/mm (0.003–0.005 in./in.), depending on mold size and design.
- Maintain mold temperature around 15°–40°C (60°–100°F) for best cycle time. Mold temperature may be increased to enhance fill, but avoid mold temperatures above 55°C (131°F) because part sticking may occur. In this case, a lubricant such as zinc stearate may be used at approximately the 0.1% level, which does not affect optical properties. Higher levels may cause yellowing.

DON'T

- Use high back pressures except when including regrind material or color concentrates.
- Use large mold vents since excessive gas is not produced.
- Small, conventional vents of 0.03–0.13 mm (0.001–0.005 in.) are adequate. Discard scrap material. If it was initially dried and processed properly and is free of contamination, it may be blended with virgin material up to a level of 25%.
- Rely on visual inspection to judge the physical properties and quality of a molded piece. Test it under exacting conditions.

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