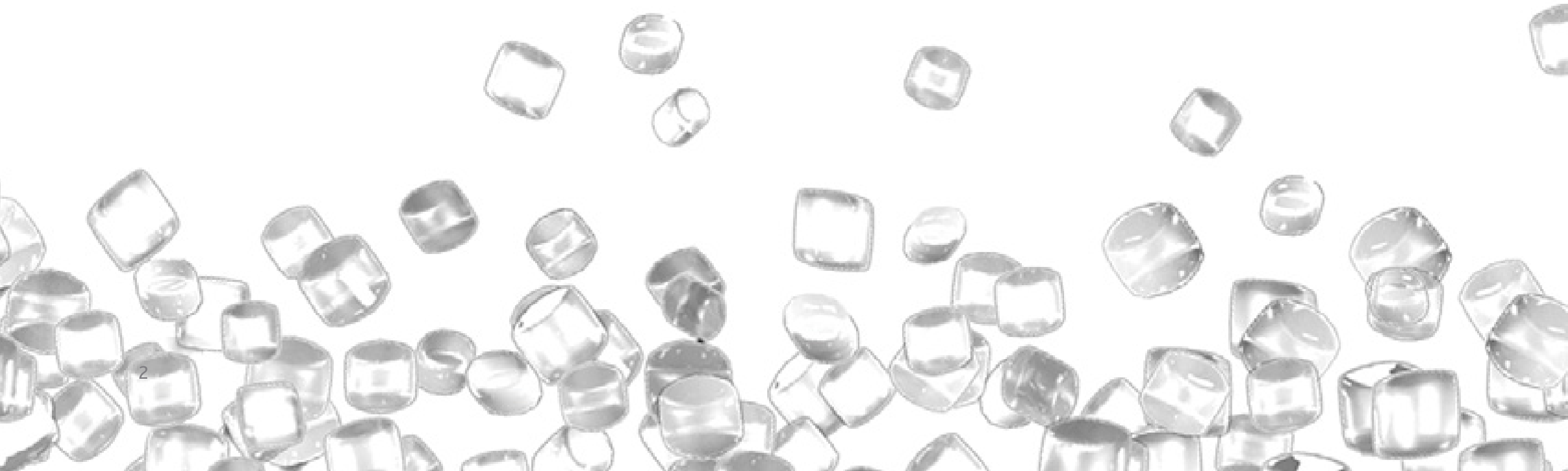
EASTMAN

Eastman **TRITAN**™
copolyester

Medical devices
processing guide

Contents



Eastman **TRITAN**™
copolyester

Medical devices processing guide

Eastman Tritan™ copolyester offers features and benefits—excellent impact strength, chemical resistance, dimensional stability, and low shrinkage—that make it suitable for use in a broad variety of commercial applications. Parts produced with Tritan are aesthetically appealing in clear and tints or with molded-in color. When combined with proper part design, suitable mold design, and recommended processing parameters, the parts provide excellent value.

This molding manual will assist designers and molders in producing the best parts from Tritan while enhancing the ease of molding.

Tritan offerings continue to expand, providing a range of processing and performance parameters that meet your needs. Table 1 is a material selection guide which lists possible medical device applications of some key formulas.

Table 1

Material selection guide							
Application	MX710	MX711	MX730	MX731	MX810	MX811	MXF121
Blood contact	●	●	●	●	●	●	
Fluid management components	●	●	●	●	●	●	
Electronic housings							●

Part and mold design

Moldability as well as product performance can be enhanced by proper product design features. Good design for moldability includes providing reasonable flow length, appropriate weld line location, moderate injection pressures, minimum clamp requirements, minimum scrap rate, easy part assembly, and minimal or no secondary operations such as degating, painting, and drilling. Good design helps minimize molded-in stress, flash problems, sink marks, and many other common molding defects that reduce quality or productivity.

Part design

The ability to fill a mold with reasonable injection pressures is greatly influenced by the wall thickness of the part. Gate location and wall thickness can be varied to achieve the best balance of part weight, clamp, tonnage requirements, and weld line location.

A part can rarely be designed with a uniform wall thickness because of such features as ribs and bosses. When wall thickness is not uniform, it affects moldability, molded-in stress, color uniformity, and structure. Effects are most pronounced if the change is not gradual, especially when the flow path throttles down from thicker to thinner cross sections, creating excessive stress and shear at the transition point.

If a part has a nonuniform wall thickness, the gate should be located in the thickest wall section to fill this section first. This will result in more efficient filling and packing at the molding pressures and temperatures selected, improving overall processability while avoiding many common molding defects. Figures 1 and 2 are examples of spiral flow.

Figure 1 Spiral flow data at midrange temperatures (3.2-mm [0.125-in.] wall)

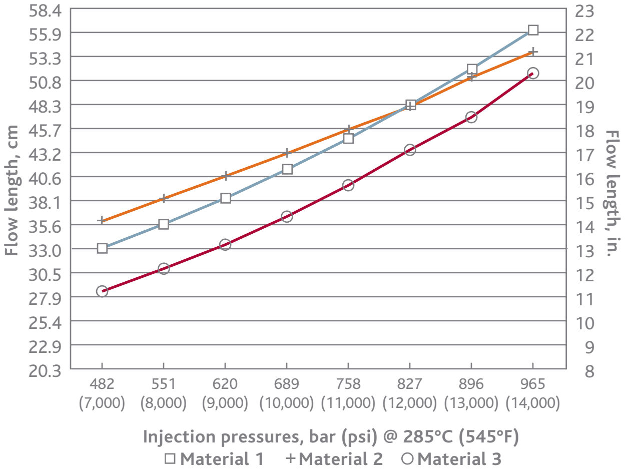
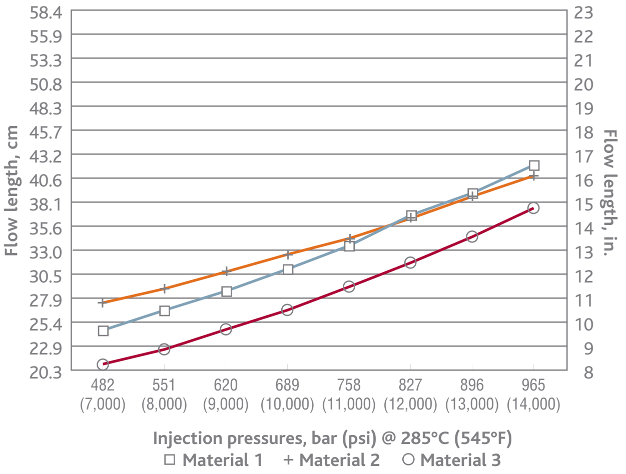


Figure 2 Spiral flow data at midrange temperatures (2.8-mm [0.110-in.] wall)



Mold filling analysis

Computer-aided mold filling analysis is particularly useful in part and tooling design. Flow patterns can be observed to determine the existence of flow imbalances which can be corrected by adjusting wall thickness, placement of flow leaders, or both. Imbalanced fill can result in underpacked areas of stalled melt-flow fronts that become cool and difficult to restart; these conditions cause molded-in stress and nonfill conditions.

The following mold filling analysis accepts data for the parameters shown under "Typical inputs" and is capable of supplying the information shown under "Typical outputs."

Table 2

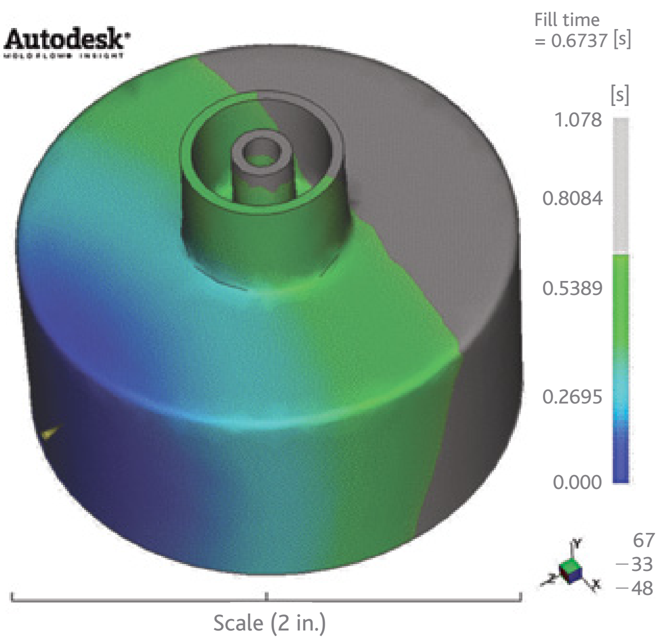
Typical inputs	Typical outputs
System geometry (typically supplied as 3D computer model files)	**Graphical outputs available from simulation**
Parts	Fill pattern (weld lines, air traps, flow front, hesitation)
Runners	
Gates	
Cooling line arrangement	Fill-pressure requirement
Resin properties	Clamp tonnage prediction
Mechanical	
Thermal	Flow front temperature
Viscosity	Time required to freeze
Pressure, volume, temperature data	Volumetric shrinkage
Processing conditions	
Mold temperature	
Melt temperature	
Injection profile	
Packing profile	
Cooling time	

Using this method, if a factor in the input is changed, the effects on moldability can be seen quickly. For example, when a gate location is changed, the differences in fill patterns, weld lines, pressures needed, and other characteristics of the molding process are shown.

An example of a mold filling simulation output predicting the fill pattern for a part is shown in Figure 3.

Eastman's analyses also make extensive use of its knowledge of thermal conductivity, specific heat, melt density, and rheological characteristics of the materials involved. All of these values vary with temperature and must be accurately known for the complete range of processing temperatures.

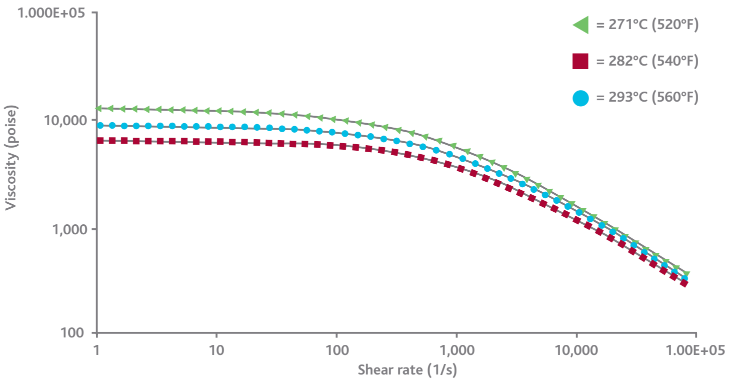
Figure 3 Computer-aided mold filling analysis image



Mold filling analysis is critically dependent on the viscosity/temperature/shear rate relationship of the molten plastic. This relationship is often shown graphically as a set of curves of viscosity versus shear rate. Some of the curves for Eastman Tritan™ copolyester MX711 are shown in Figure 4.

Molten plastics typically exhibit non-Newtonian flow behavior; that is, their resistance to flow does not vary linearly with the pressure applied. The material is said to be shear-thinning if doubling the pressure results in more than twice the flow rate. The viscosity of Eastman polyester plastics has been determined for shear rates ranging from approximately 200 to 10,000 sec $^{-1}$. The values of this property affect the injection pressure required and the clamping force needed in the injection machine, as well as other factors.

Figure 4 Shear rate/viscosity curves for Eastman Tritan™ copolyester MX711



Ribs and radii. Ribs can be added for stiffness and combined with bosses for fastening and other functions. To minimize sinks on the opposite side of the part, rib thickness should generally be about half the wall thickness of the part. Locate ribs so that appearance of even very minor sinks will not compromise part appearance.

When possible, ribs should run parallel to flow from the gate to minimize air entrapment and high shear when filling the part. Ribs and bosses should have radii where their walls meet the part walls. (A minimum radius of 1.14 mm [0.045 in.] or 0.4–0.6 times the wall thickness is suggested.) Radii add strength by removing notches and aid mold filling by streamlining the flow channel.

Part removal. A good design should allow for easy part removal and simplicity of operation by providing adequate mold-open clearances, ample draft angles, and ample ejectors. These features will enhance moldability by promoting short cycles and minimizing variability in the molding cycle. Also refer to "Sprue design" and "Drafts for cores and cavities" on pages 8–9.

Mold design and construction

Tool steels and mold construction

There are several factors to consider when selecting steel for the mold, including wear resistance, toughness, machinability, ability to be polished, and dimensional stability. The steels most often used are P20, H13, and S7.

- Core and cavity steels
  - P20 steel is supplied prehardened at a Rockwell hardness (R scale) of 30 to 32, which eliminates the need for heat treatment. P20 will polish to a very high finish, but rust-preventive greases will be required during shutdowns to preserve the finish; otherwise, plating will be necessary. Plating can be an impediment during repairs. The thermal conductivity of P20 is better than that of H13, 420, and S7, but its conductivity could eventually be impeded by cooling channel ID corrosion. P20 costs less than H13 and 420.
  - H13 steel typically requires heat treatment for more hardness and durability. H13 has less toughness and thermal conductivity but higher wear resistance than P20. Because of its greater hardness, parting lines in H13 hold up longer than those of P20. With reduced thermal conductivity, increased cooling should be considered. H13 can also rust if not properly protected during use.
  - Although 420 stainless steel has lower thermal conductivity than H13, it offers rust resistance on the polished surface and cooling channel ID that is not available with P20 or H13. Heat treatment similar to that of H13 is required for 420SS. Some suppliers also have a 414SS prehardened the same as P20 at an R scale of 30 to 32, which eliminates the need for heat treatment.
- Slides and lifters
  - S7 tool steel is often used for hardened slides and lifters. Wear plates and gibs are often constructed from O1, O6, and A10. Bronze or bronze-coated (Lamina™ bronze) plates are also used adjacent to sliding surfaces.
- Inserts
  - Inserts, which need to have superior thermal conductivity, may be constructed from a tooling material such as Ampco™ bronze alloy 945 or equivalent. This alloy is supplied in an RC of 31.

Because processability is dependent on the tool, it is necessary to consider material options and toolmaker recommendations carefully. Tool investments will pay huge dividends in products.

Gating

Gates are of several basic types: (1) sprue gating directly into the part; (2) fan gates; (3) flash gates; (4) edge gates; and (5) hot runners with custom-designed gates. The size and appearance of the finished part must be considered in selecting the type and location of gates.

Considerations for gate location(s) include the following:

Minimizing flow length. Minimum flow lengths are typically made possible by locating the gate near the center of the mold. This minimizes pressure needed to fill the cavity, optimizes wall thickness necessary for easy molding, and reduces part cost.

Weld line (knit line) location. Although some polymers have relatively low-visibility weld lines, gate location does determine where weld lines will form. This should be considered in advance.

Minimizing gate blush. Gate blush is an aesthetic defect typically associated with high shear stress during the filling stage of the injection molding process. In highly transparent resins such as Eastman copolyesters, gate blush can appear as a white hazy area surrounding the injection location.

Although adjusting processing variables such as increasing melt temperature or slowing injection speed can often be used to minimize shear stress and blush, gate design is also a factor. Larger gate sizes have less shear stress, reducing the tendency for blush. Smooth transitions around corners and thickness changes throughout the runner and gate system are also beneficial in minimizing blush.

Wall thickness. If it is necessary to make the wall thickness of a part nonuniform, gating should be into the thickest area. Transitions should be gradual and smooth from thick to thin.

Streamlining the flow path helps maintain low shear. No sharp corners or sudden changes in thickness should be allowed. If a transition is needed from a thick sprue or runner to a thin wall, the change needs to be smoothly radiused over the available distance. To ease material flow, runners and gate edges should be round rather than trapezoidal or square. It is good practice to gate into areas where the flow path is continuous and smooth rather than into notches or ribs.

Runners

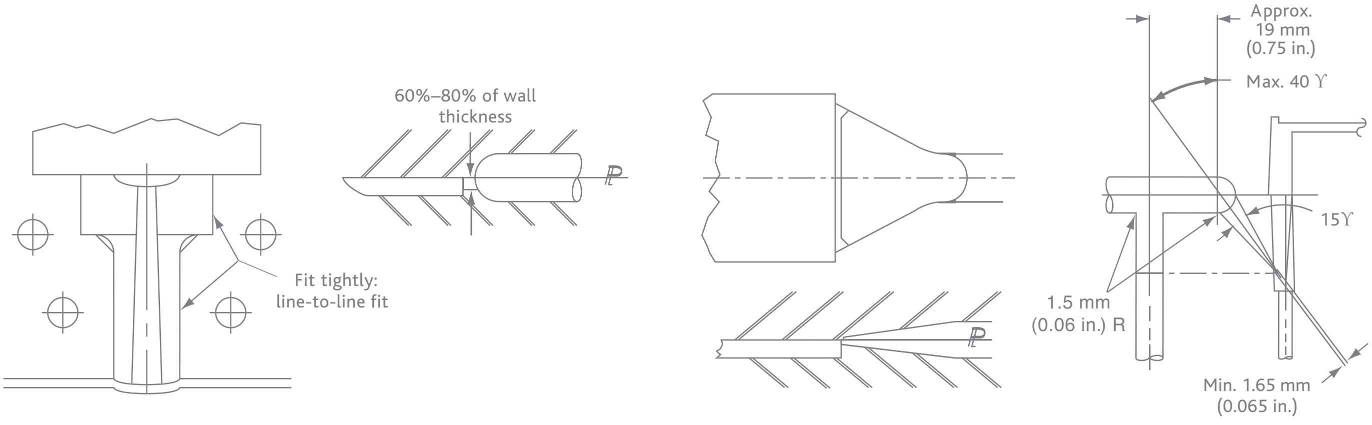
Full-round runners provide maximum flow with minimum mold contact, decreasing pressure drop through the runner system. Square, half-round, and trapezoidal runners are less effective because they only allow flow approximately equivalent to that of a full-round runner that could be placed inside them. Runner size is based on part size, flow length, material viscosity, mold and melt temperatures, and gating. Cold slug wells should be provided where runners make right-angle turns.

Cold runners. In general, cold runners of about 13-mm (½-in.) diameter are suggested for parts of 2.3 kg (5 lb); for parts under 1.8 kg (4 lb) with relatively short runners (25–125 mm [1–5 in.]), a diameter of 8–9.5 mm (⁵⁄₁₆–⅜ in.) is suggested. Figure 5 shows typical cold runner gate designs.

Hot runners. Hot runner and valve-gate systems have been used successfully with amorphous copolyesters and alloys. *However, the selection of a suitable hot runner system can vary greatly depending on the size of the part, polyester formulation, and part design. Therefore, it is critically important that runner design and selection be discussed jointly by the molder/end user, tool builder, hot runner supplier, and Eastman to arrive at the appropriate runner system design to be used.*

Necessary design features include those noted on the next page of this guide.

Figure 5 Typical cold runner gate designs



Typical sprue gate **Edge gate** **Fan gate** **Sub gate**

Uniform heating and good heat control. Temperature control at the tip is extremely important. The tip must be hot enough to heat the material so that it is soft and flowable, but the adjacent mold cavity must be below 55°–65°C (130°–150°F) to prevent material from sticking to the hot steel and prevent the formation of heat sink marks in the molded part. This is best accomplished by the gate orifice being an integral part of the cavity steel rather than the hot runner system being an insert projecting through the cavity into the part. When the gate is in the cavity, cooling channels (drilled water lines or annular-shaped passages) can be incorporated to provide the cooling needed for the cavity in the gate area; some hot runner suppliers offer gate cooling inserts.

The hot probe-style drop needs to have its own thermocouple, heat source, and control to allow regulation of the temperature as needed. The thermocouple should be located near the probe tip for accurate temperature control with minimal variation.

Elimination of holdup spots. The flow channel for the plastic should be streamlined and uninterrupted. Any crevice or pocket where material can collect and degrade will probably cause degradation.

Minimization of shear heating. The diameter of the flow path needs to be large enough to minimize the shear heating that can be caused by sharp corners or edges in the flow path at the gate or elsewhere. Mold filling analysis can show shear heating and indicate potential problems during the design stage.

Mold temperature control

A benefit of the low mold temperature needed for amorphous copolyester polymers and alloys is shorter molding cycle times. To ensure proper cooling, numerous channels are needed. There should also be an adequate supply of temperature-controlled water. Ribs and deep-draw areas need special attention during mold design; bubblers or baffled coolant drops are used to avoid local hot spots that can lengthen the molding cycle or reduce part quality.

Sprue design

Proper sprue design is important for good molding and easy removal of the part from the mold. For many materials, the sprue has a taper (including angle) of 40 mm/m (½ in./ft). Sprues should be as short as possible, preferably less than 75 mm (3 in.). Cooling is important and can be provided by ample cooling channels near the sprue bushing. Sufficient metal-to-metal (thermal) contact between the bushing and the cool steel of the mold is also required. Figure 6 illustrates relative dimensions of drilled coolant lines, and Figure 7 illustrates baffled coolant drops.

Figure 6 Drilled coolant lines

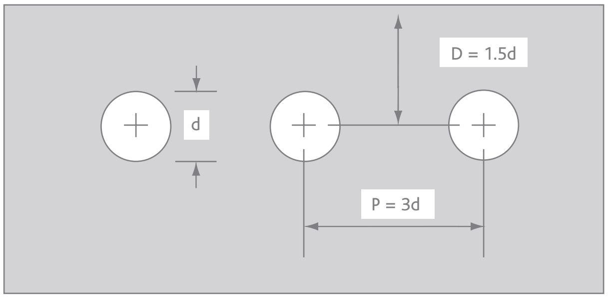
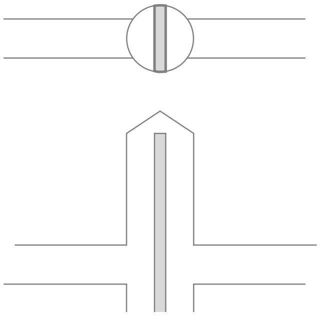


Figure 7 Baffled coolant drops



Make drop larger than supply feeder so that the area of half the drop does not restrict feeder. Do not oversize to point of reducing velocity whereby laminar flow can result.

Use of sprue bushings made from a tooling material such as Ampco™ 940 bronze, which have improved thermal conductivity over steel bushings, is strongly suggested. Most metric bushings are either 1° or 1.5° per side ID taper; the 1.5° per side taper would be the bushing of choice for amorphous copolyester and alloy resins. The ID sprue surface should be polished sufficiently to remove all radial machining grooves and any undercuts. If sprue sticking still occurs, then a bushing with an included angle taper of 60 mm/m (¾ in./ft) should be tried; this is especially beneficial when steel bushing material is used. Water-cooled sprue bushings are available from some suppliers.

The diameter of the small end of the sprue should be larger than the nozzle opening by approximately 0.8 mm (1⁄32 in.).

Hot sprues can be used for amorphous copolyesters and alloys. As with hot runners, the keys to proper design are low shear, good cooling at the part or sprue/runner end, uniform heating, and good temperature control.

Venting

Molds should be well vented around the perimeter of the part and at other areas where a flow front meets a wall or another flow front. Typical depth of a vent is 0.03–0.04 mm for approximately 6 mm (0.001–0.0015 in. for approximately 0.25 in.) opening to a larger channel vented to the atmosphere.

Drafts for cores and cavities

A draft of 1° is typical for parts molded with Eastman amorphous copolyesters and alloys, although lower drafts can sometimes be used with well-designed tools. If the surface of the mold is textured, an additional 1° to 1.5° should be allowed for each 0.025 mm (0.001 in.) of texture depth.

Zero-draft situations should be avoided, and drafts lower than the values given should be used only when absolutely necessary.

Cores and tools should be polished in the direction of draw. Cores should be cooled to avoid sticking, and ample ejectors should be used.

Mold shrinkage and warpage

Key factors in minimizing warpage include uniform wall thickness and consistent mold temperatures. A uniform wall promotes even flow, minimizes shear heating, reduces molded-in stress, and tends to minimize warpage.

A uniform mold temperature helps ensure even heat transfer from both wall surfaces. This will leave the part in a balanced condition, provided the wall thickness is uniform. The important factor is control. The mold should be designed for adequate control of the temperature in the range required for the material being processed. This will not only decrease the amount of residual stress but also permit reduction of cycle time.

Processing information

Some of the parameters to consider in choosing a machine for molding Eastman Tritan™ copolyester are barrel and melt temperatures, mold temperatures, fill speed, screw speed (rpm), pack and hold, cushion size, back pressure, decompression (suck back), screw and barrel design, purging, and annealing.

Processing

Barrel and melt temperatures

Consistent part production requires attention to all phases of the injection molding process. Processing conditions should be optimized to ensure material integrity and maximum part performance. Some recommendations for processing Eastman Tritan™ copolyester:

- Processing at the optimal processing temperature and minimum residence time in the machine will assist in maximizing physical properties.
- Well-dried material is the key for shot-to-shot uniformity. Engineering materials such as Tritan can suffer degradation at their processing temperatures due to hydrolytic degradation.
- Normal processing temperatures are in the range of 282°C (540°F) plus or minus 5°–10°C (10°–20°F) measured by air shot. Parts run at faster cycle times utilizing higher barrel capacity (50%–80%) can be run at the higher end of the melt temperature range. Conversely, when parts are molded with long cycle times utilizing a minor amount of the barrel capacity (10%–25%), the processor should strive to run Tritan at the lower range of the proposed melt temperature.
- For Tritan MXF121, special attention should be given to making certain that adequate mixing of the resin components occurs during plastication. Many parts made from this material can be quite large, so maintaining the appropriate shot-to-barrel size ratio is important to allow for air escape and homogeneous melt preparation.
- A flat temperature profile setting is normally used when shot size is approximately 50% of barrel capacity; a barrel with a three-zone system might have the following settings:

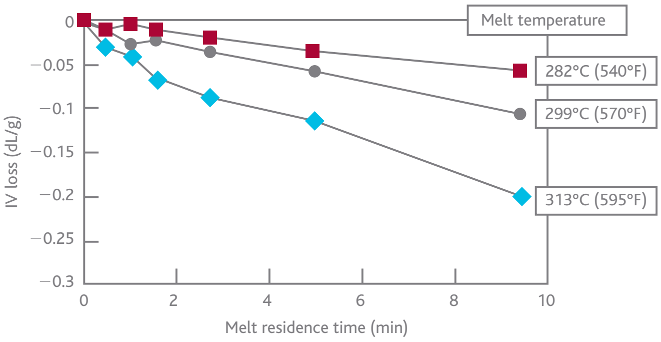
Table 3

Rear zone	282°C (540°F)[a]
Center zone	282°C (540°F)[a]
Front zone	282°C (540°F)[a]
Nozzle zone	282°C (540°F)[a]
Hot runners	282°C (540°F)[a]
Actual melt temperature (purged on cycle)	282°C (540°F)[a]

[a] *Since each machine is different, the barrel set temperatures might need to be set as much as 10°–20°C (20°–40°F) lower than the targeted melt temperature due to shear heating. It is good practice to determine the actual melt temperature, temperature inside machine nozzles, and temperatures inside hot sprues and runners using a pyrometer. Also, it is important that the casting around the throat of the injection molding machine is cooled to provide optimum pickup of the material.*

- In special situations, Tritan does have a wider processing window (depending on the process) ranging from approximately 260°C (500°F) where flow and screw recovery become stiff to approximately 304°C (580°F) where splay may begin at 10-minute melt residence times. A good setting is generally 282°C (540°F) while targeting a melt residence time (screw and hot runner time) of 5–6 minutes. Figure 8 shows the effect of melt residence time on material integrity over a range of melt temperatures. Inherent viscosity (IV) loss in Tritan as a function of melt residence time over a melt temperature range is shown. IV is a commonly used molecular weight indicator in copolyesters.

Figure 8 Inherent viscosity (IV)



Mold temperatures

Good temperature uniformity through the mold and good temperature control to a set point are key to successful molding.

- Actual mold surface temperatures ranging from 60° to 66°C (140° to 150°F) produce the best low-stress parts. Recall that the actual water temperature going into the mold may be lower than mold surface temperature if heat transfer is relatively slow.
- Eastman Tritan™ copolyester requires colder molds than some other plastics, so preparing cooling ahead of time pays dividends in cycle time and processability. High mold temperatures, even in small areas of the mold, can cause sticking. Ample mold cooling channels, uniform wall thickness design, good cooling of pins and thin steel areas, good cooling near hot spots such as sprues or hot runners, insulating areas around hot runners, good water supply with few flow restrictions, and Thermolators® for exact setting control of water temperature all assist in generating fast-cycling parts with good surface appearance.
- With good cooling as outlined above, the cooling portion of the cycle can be minimized to a point where the part is solidified and easily ejected while the larger-diameter sprue is often still soft and rubbery.
- Additional cooling could be needed to prevent sprue sticking. Review the mold construction guidelines on page 13 for additional information.

Fill speed

- Fill speeds used for Tritan are slower than typical plastics. Machines with fill-speed profile capability are recommended. Where fill-speed profiling is available, starting the fill at a very slow speed such as 13 mm (0.5 in.) per second for the first 5%–15% of the shot, increasing to 43 mm (1.7 in.) per second, and then slowing to 23 mm (0.9 in.) per second is often successful. The slower initial fill speed minimizes gate blush. When direct sprue gating into the part is used, a moderate to fast fill rate, such as 38–56 mm (1.5–2.2 in.) per second, is suggested.
- Gate geometry is also very important to part appearance near the gate. If the gate or runner has sharp corners or other nonstreamlined features in the flow channel, these may need to be radiused to reduce blush near the gate. Gate thickness as well as speed can influence gate blush. Gate thicknesses less than 1.1 mm (0.045 in.) are not suggested for most gate types.

Screw speed (rpm)

Plastication should be slowed to the minimum speed necessary to recover the screw during part cooling and sit at the rear position only 2–5 seconds before the mold opens. This minimizes high-speed shear and tends to make the melt more uniform. In processing Tritan, lower rpm can make screw recovery more steady and consistent.

Pack and hold

Where direct sprue gating into the part is used, longer hold times in combination with lower hold pressures may be necessary. If a void develops at the base of the sprue, the sprue has a tendency to stick in the mold, separating at the part. Packing out the void strengthens the sprue such that it will now release with the part. Having long hold times of 8–12 seconds and lower hold pressures of 34–52 MPa (5,000–7,500 psi)[1] will feed material to the sprue to fill the void while not overpacking the sprue. Overall cycle time does not have to be extended if the cooling time is decreased by the same amount the hold time is increased. Sticking can also happen with a conventional runner at the junction of the runner and sucker pin. Again, if the sprue sticks in the mold, utilizing the same methodology will help solve the problem.

Cushion size

Cushion size should be at the absolute minimum to assure the screw does not hit bottom and the pack and hold pressures are getting into the part. The cushion left at the end of the pack and hold is typically 5–10 mm (0.2– 0.4 in.), depending on machine size and injection speed. Larger cushions can add to holdup time in the barrel and aggravate degradation. If the screw continues to move forward at the end of the shot when adequate time is given to come to a stop, this is a sign of a leaking check valve. A leaking check valve may also cause short shots and shot-to-shot variability.

[1] *Note that these pressures are actual melt pressures, not gauge pressures (often gauge readings are 1⁄10 actual pressures depending on machine and barrel).*

Back pressure

Back pressure is usually kept to a minimum of about 10 MPa (1,500 psi).[1] However, to improve melt uniformity (and mix concentrates), increase melt temperature, or get rid of air entrapment (air splay), back pressure can be increased gradually to as much as 15.5 MPa (2,250 psi).[1] High back pressures can aggravate drooling into the mold and require additional decompression. Especially for largest-part molding, high back pressure may be helpful in better mixing of the material.

Decompression (suck back)

In general use, very little or no decompression occurs. Decompression tends to pull air back into the nozzle, causing splay in the next shot. Very small amounts of decompression can be used to reduce drool if needed.

Screw and barrel design

Eastman Tritan™ copolyester has been processed in a wide variety of general-purpose screws with compression ratios in the 2.8:1 or 3:1 range and L/D ratios of 18–22:1. The transition zone should have a gradual transition (typically 4–6 diameters) so that the high shear heating of a sudden transition is avoided. Screws should be chosen to be compatible with the hardness of the barrel material to minimize wear as with any plastic material. Unfilled materials, such as Tritan, are generally very mild on screw wear. Corrosion of barrel and screw parts is not expected with Tritan.

Purging

For purging when going from other polymers to Eastman Tritan™ copolyester:

- The material most effective in purging is a polymer similar to the material to be run. Polyethylene and polypropylene should be avoided because they can mix with the new material and cause streaks for extended periods of time. Use caution and refer to the manufacturer's recommendations for the material used in the previous run.

For purging when going from Eastman Tritan™ copolyester to other polymers:

- Purge with acrylics, polystyrene, commercial purging compounds, or the polymer to follow Tritan.

Purging with other materials is not needed when Tritan is going to be run again after a shutdown. For a machine shutdown, such as a weekend, simply shut off the pellet feed, run the screw empty, and turn off heat to the barrels and hot runners. Start up again with barrel heat which takes the longest, then turn on hot runners with just enough time to reach set point at the same time as barrels. When set points are reached, start right away to avoid sitting and cooking the polymer. For short shutdowns, such as during brief repairs, if the machine is sitting at set temperature longer than about 10 minutes, it's generally suggested to purge (air shots) the barrel contents and restart molding.

Table 4 Summary of the recommended drying and processing conditions for injection molding Eastman Tritan™ copolyester

Drying conditions	
Drying temperature, °C (°F)	88 (190)
Drying time, h	4
Dryer air dew point, °C (°F)	<229 (<220)
Processing temperatures	
Zones °C (°F)	
Rear	Set barrel temperatures to reach target melt temperature, up to 10°–20°C (20°–40°F) below target depending on shear heating
Center	
Front	
Nozzle °C (°F)	282 (540)
Hot runners °C (°F)	282 (540)
Melt temperature °C (°F)	282 ± 10 (540 ± 20)
Mold temperature °C (°F)	60 (140)
Machine conditions	
Injection speed	Slow
Screw speed (rpm)	Minimum
Pack and hold pressure (MPa)	35–50
Cushion (in.)	0.2–0.4
Back pressure (MPa)	10–15

---

[1] *Note that these pressures are actual melt pressures, not gauge pressures (often gauge readings are ⅒ actual pressures depending on machine and barrel).*

Annealing

When mold surface temperatures are maintained in the suggested ranges, relief of residual stress through annealing is typically unnecessary. This recommendation should be evaluated in individual cases where residual stress is of particular concern.

Mold construction

The following guidelines minimize cold sprue sticking or sticking around the gate, reduce cycle time, and open the processing window.

General guidelines

- Design molds to maintain the desired uniform mold surface temperature of 60°–66°C (140°–150°F) even when run at aggressive cycle times.
- Use water-line spacing of 50–64 mm (2–2.5 in.) between center lines.
- Air poppets should be offset from the center line of the sprue or gate as far as possible.
- Balanced runner systems are suggested so that temperatures and pressures are similar for all cavities and flow is simultaneous to all cavities.

Cold runner mold construction guidelines for Eastman Tritan™ copolyester

- Taper to be 3° minimum (including an angle) on the sprue bushing.
- Shorten the sprue bushing "L" dimension to less than 75 mm (3 in.) in length.
- Orifice size of the sprue bushing where the sprue bushing meets the nozzle should be 4–7 mm (5⁄32–9⁄32 in.) in diameter. Larger parts will need orifice diameters of 7 mm (9⁄32 in.) while smaller parts will need only a 4-mm (5⁄32-in.) diameter orifice.
  - For example, a sprue bushing for a medium-sized part should have a length of 75 mm (3 in.) or less and a sprue bushing orifice diameter of 5.5 mm (7⁄32 in.).
- The sprue bushing is to have a high polish in the sprue area.
- Increase cooling around the sprue bushing—suggest upper and lower water-line circuits.
- Maintain good surface contact between the sprue bushing and mold surface.
  - Suggest line-on-line interference fit.
  - Surface contact is to be on the head of the sprue bushing as well as the shaft.
- In cases where aggressive molding cycles are desired, substitute an alloy sprue bushing for the steel sprue bushing. Alloy sprue bushings are fabricated from raw materials that enjoy significantly better thermal efficiency than traditional steel sprue bushings.

Hot runner mold construction guidelines for Eastman Tritan™ copolyester

- Cleanly separate the hot and cold areas of the mold with good insulation systems so that melt is uniform at 282°C (540°F) and the well-cooled mold is maintained at its uniform surface temperature of 60°–66°C (140°–150°F), especially including the area around the gate.
- Ideally, the melt should be maintained at the same temperature generated at the discharge of the screw all the way through the machine nozzle, mold sprue, hot runner manifold, and hot runner drops and tips.
- In general, the preferred hot runner gating system is a water-jacketed valve gate with an insulator to eliminate dead spots.

Nozzles

Select nozzles with the minimum length needed to extend into the mold. General-purpose nozzles of uniform bore or larger-diameter nozzles that use generous radii to gently reduce diameters at the exit end are preferred. The inside diameter of the nozzle should be very close to that of the sprue end but just slightly smaller so that the sprue can be pulled. Nozzles with inside diameters of 5–8 mm (3⁄16–5⁄16 in.) are typical for smaller parts; those for larger parts should have a 9.5 mm (3⁄8 in.) or larger diameter.

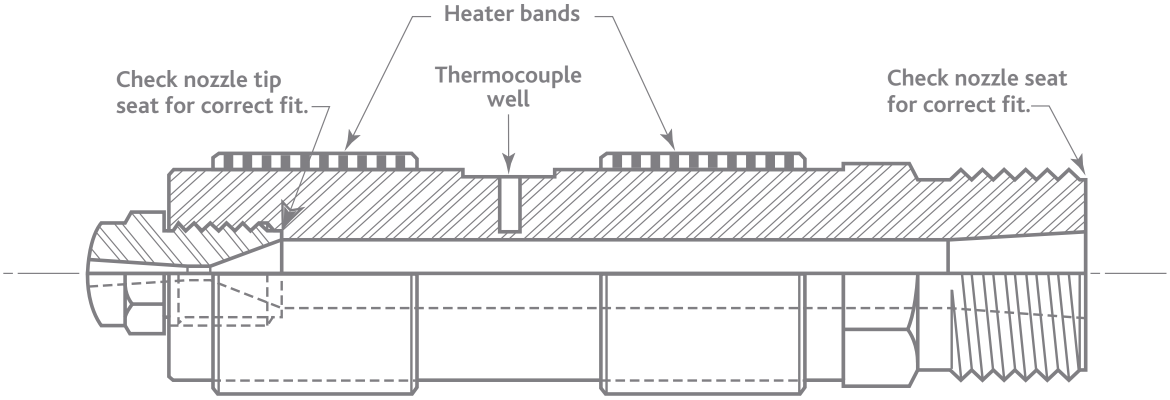
Good temperature control of the nozzle is important. If a thermocouple at the threaded end of a long nozzle is controlling a heater band or bands at the other end, temperatures at the heater band end can be more than 55°C (100°F) higher than the thermocouple is able to sense. This can be checked by inserting a needle pyrometer to different depths in the nozzle opening. The remedy is to either reduce the set point of the controller or, preferably, use a nozzle fitted with a thermocouple in the center of its length with heater bands located uniformly on both sides as shown in Figure 9.

Long nozzles may require more than one thermocouple/controller/heater band along their length for uniform heating. Nozzles with gas-charged heat pipes have been used successfully to heat the full length of long nozzles more evenly. Temperature control problems in the nozzle show up as appearance problems at or near the gate. Nozzles with a removable tip require special attention to verify that the tip bottoms out on the shoulder below to prevent a dead space where resin can degrade; if this happens, black specks can form and reenter the melt stream.

Periodic inspection

The screw, check valve, and nozzle assembly should be taken apart, cleaned, and inspected periodically to measure wear and look for cracks or any other spots where material can collect and degrade. Small cracks or unseated threads can be big enough to cause streaking or degradation

Figure 9 Injection nozzle



Nylon configuration suggested for copolyester.

Drying

Drying is necessary.

All polyester resins readily absorb moisture. Desiccant dryers must be used to dry the pellets prior to processing in the injection molding machine. A typical desiccant dryer is shown in Figure 10.

Drying is an absolute necessity to prepare Eastman Tritan™ copolyester for molding. If pellets are not dried, moisture will react with the molten polymer at processing temperatures, resulting in a loss of molecular weight. This loss leads to lowered physical properties, such as reduced tensile and impact strengths.

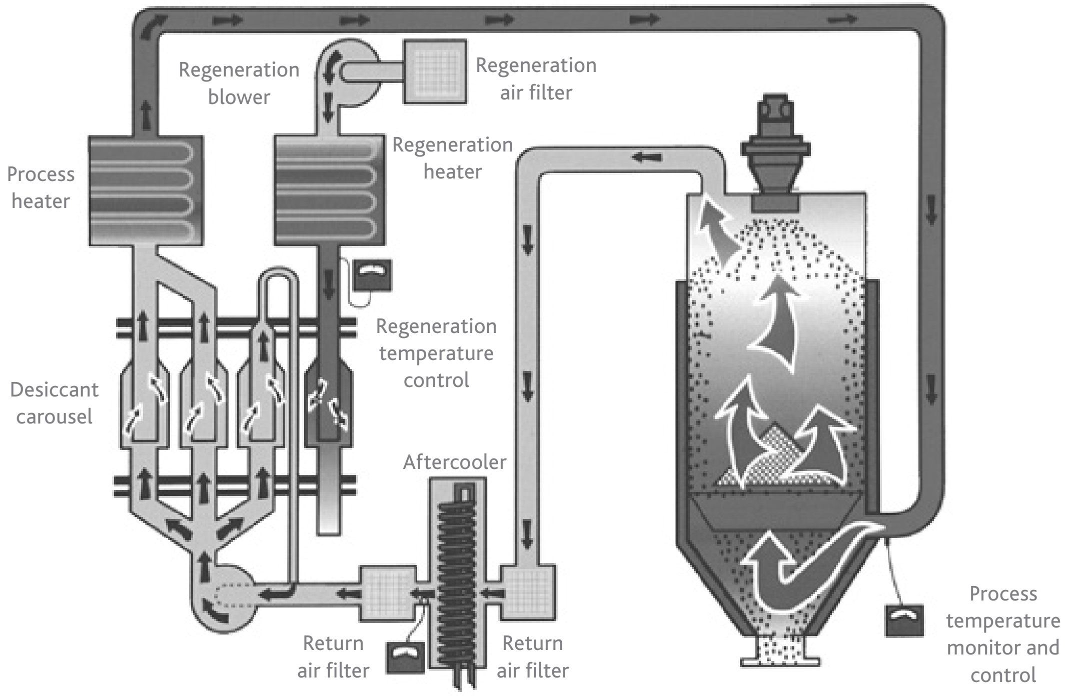
Molded parts may not show any noticeable defects, such as splay, but may still exhibit lower physical properties.

Drying equipment

Multibed desiccant dryers are recommended to properly dry the resin. These dryers have two or more desiccant beds. Dryers that have three or four beds typically have shorter start-up times due to quicker bed regeneration. Desiccant dryers are available from many suppliers. Work with your desiccant dryer vendor to select the optimum dryer for the molding job. Locating the drying hopper on the feed throat of the molding machine is preferred. Planning should include consideration for throughput rate, ease of maintenance, reliability, and low variability of the four elements necessary for proper drying (drying temperature, drying time, dryness of air, and airflow).

Tray dryers can be used only if they are supplied with air dried by a good desiccant bed system. Tray dryers with heating only (and no desiccant) do not adequately dry the pellets. Good dryers for production typically include either rotating beds or other means to keep continuous airflow through a freshly regenerated bed while other beds are regenerated offline. Tray dryers with manually charged single beds are also generally not recommended for continuous production operations.

Figure 10 Typical desiccant dryer



Drying conditions

Effective drying of Eastman Tritan™ copolyester is key to shot-to-shot consistency and optimum part performance. The following are important points to consider for proper drying of Tritan.

- Dew point: Use a desiccant type or similar drying system providing dry air at a minimum dew point of 229°C (220°F).
- Time and temperature: Dry Tritan at 88°C (190°F) for 4 hours minimum. If longer residence time in the dryers is required, such as overnight, lower the set temperature to 82°C (180°F). The inlet air temperature needs to be controlled within ±3°C (±5°F) throughout the drying cycle.

Dryer diagram and troubleshooting

Dryers require routine checking and maintenance. A mechanic who understands dryers and has the time and support to maintain them is needed. The following information is provided to help give that understanding. Dryer suppliers can help also.

Common dryer problems

- Poor airflow caused by clogged filters
- Air passing through the middle of the load rather than dispersing through the pellets, caused by unfilled hopper
- Supply/return dry air lines allowing ambient “wet” air to contaminate dry air
- Wet air contamination through loader on top of hopper
- Lack of cooldown on air returning to the bed in the absorption process. Air should be cooled below 65°C (150°F) to increase the desiccant’s affinity for moisture, thus improving efficiency. An aftercooler is required when drying some resins.
- Reduced desiccant effectiveness caused by worn-out or contaminated desiccant
- Nonfunctioning regeneration heater and/or process heater
- Blower motor turning backwards
- Airflow not being shifted when controls call for bed change; one bed stays in process continuously

Elements necessary for proper drying

A discussion of the four elements necessary for drying plastics follows.

1. Drying temperature

Resin must be dried at a specific temperature. Air circulating through the hopper is heated by the process heater or afterheater. Air temperature should be measured at the inlet to the hopper and controlled at the recommended drying temperature for a given resin. Exceeding this temperature will cause premature softening or melting of pellets to the point of sticking together, causing failure to feed freely to the bottom of the dryer for unloading. Drying at temperatures below the recommended set point will result in inadequate drying. When the controlling thermocouple is located away from the hopper, the set point may need to be raised to offset heat loss from the air during transport to maintain the desired hopper inlet temperature. Check the temperature over several cycles of the process heater. If the actual temperature overshoots the set point, adjust the set point accordingly to avoid overriding temperatures. Drying temperature should be held constant within ±3°C (±5°F). Insulated supply hoses and hoppers make drying much more effective and save energy costs.

It is also important to maintain air temperature (at least 205°C [400°F]) in the regeneration loop of the dryer. The regeneration loop is a separate system from the process loop, so the presence of hot air in the process loop does not guarantee that the regeneration loop is functioning.

2. Drying time

Pellets to be dried need to be in the hopper at the proper conditions. If the dryer is turned on from a cold start, it must warm up to the proper temperature and the dew point of the air must be reduced to 230°C (220°F) or below before drying time can be counted. Choosing the hopper size is critical; only when the hopper size is adequate for the rate of processing will the proper residence time in the hopper be possible. For example, if a 454-g (1-lb) part is being run on a 1-minute cycle, then 27.2 kg (60 lb) of dry material will be needed each hour. If 6 hours is required for drying, then at least 164 kg (360 lb) of material must be in the hopper continuously (27.2 kg/h × 6 h [60 lb/h × 6 h]). The hopper should be built so that plastic pellets in all parts of the hopper will move uniformly downward as material is removed from the bottom. Funneling pellets down the center of the hopper while pellets near the outside move more slowly will result in inadequate drying.

In routine operation, drying time is maintained by keeping the hopper full. If the hopper level is allowed to run low, residence time of the plastic in the hopper will be too short and the material will not be adequately dried. For this reason, and to compensate for less-than-perfect plug flow through the dryer, the hopper should be larger than the exact size calculated.

3. Dryness of air

Dry air comes from the desiccant beds in the closed air-circulation loop of the dryer/hopper system. Desiccant beds must be heated and regenerated before they can dry incoming processed air. After regeneration, it is beneficial to cool down the regenerated bed with closed-loop (previously dried) air as opposed to ambient air.

Returning processed air from the top of the pellet hopper is filtered before it is blown through the desiccant bed and onto the heater and hopper. Dryers used for polyesters should be equipped with aftercoolers to cool the returning processed air. Air temperature should be below 65°C (150°F) to increase the desiccant's affinity for moisture, improving efficiency.

The desiccant in the beds is typically a very fine claylike material in pea-size pellets. It slowly loses its usefulness and must be replaced periodically—usually about once a year. Use of plastic with a high dust content (such as regrind) or materials containing certain additives will reduce the life of the desiccant by coating the pellets or saturating them with a nonvolatile material. Good filters can help extend the life of the bed and the heater elements.

Air dryness can be checked by dew point meters, either portable or installed in-line in the dryer (built-in dew point meters and alarms are the wise choice for polyesters). These meters give a direct reading of the dew point of the air tested. When the dryer has rotating beds, the meter must run long enough for all beds to be checked. Each bed can normally be online for 20–40 minutes or longer; a new bed should rotate into position before the dew point rises above 230°C (220°F). (Also see the "Moisture measurement" section.)

4. Airflow

The usual airflow rate requirement for drying is 0.06 cubic meter of hot, dry air per minute for each kilogram of material processed per hour (0.06 m^3/min per kg/h) or 1 cubic foot of hot, dry air per minute for each pound of material processed per hour (1 cfm per lb/h). For example, if 109 kg (240 lb) of material is used per hour, airflow should be at least 6.7 m^3/min (420 cfm). Minimum airflow to ensure good air distribution is usually about 2.8 m^3/min (100 cfm) for smaller dryers.

Airflow can be checked by in-line airflow meters, by portable meters, or much less accurately by disconnecting a hose going into the hopper and feeling the airflow.

If there are dust filters in the circulation loop, these should be cleaned or replaced periodically to avoid reduction in the airflow rate.

Moisture measurement

Dew point meters, as already mentioned, can be either portable or, preferably, built into the dryer. They measure only the dryness of the air, not the dryness of the plastic pellets in the hopper. Use of the dew point meter along with measurements of temperature, airflow, and time can give an accurate indication of whether the plastic pellets are being dried properly.

Weight-loss-type moisture meters are instruments that measure the moisture inside pellets. These meters can give a general indication of the effectiveness of the drying system in reducing the moisture level in plastic pellets. However, most are usually not accurate enough to use as a quality control method to ensure adequate dryness of polyesters to prevent degradation during process. A moisture level in the range of 0.005%–0.015% is desired, and this is determined using analytical means other than the preceding.

Dryers require routine monitoring and maintenance. A mechanic with the knowledge, time, and resources is key. Dryer suppliers can also help.

Table 5

Dryer troubleshooting guide		
Problem	**Possible cause**	**Corrective action**
High dew point (wet air)	Desiccant worn out or saturated	Dry cycle the machine or replace desiccant.
	Incorrect desiccant type	Replace desiccant with type and size recommended by dryer manufacturer.
	Regeneration heaters burned out	Replace heaters.
	Regeneration filter plugged	Clean or replace filter.
	Regeneration blower reversed	Reverse electrical connections.
	Air leaks	Check and repair auto loader seal and/or hoses to hopper.
	Beds not changing at the proper time	Reset or repair controller.
	Return air too hot	Add or repair aftercooler.
Low airflow	Dirty air filter	Clean or replace filter.
	Fan motor reversed	Reverse electrical connections.
	Hoses reversed between inlet and outlet	Connect dryer outlet to inlet at the bottom of the hopper.
	No hose clamps; hose disconnected	Connect and clamp hoses.
	Hose smashed or cut	Repair or replace hose.
Short residence time	Hopper too small	Use a larger hopper.
	Hopper not full	Keep hopper full.
	Tunneling or “rat holes”	Remove clumped material or install proper spreader cones.
Temperature high or low (or varying more than ~3°C [~5°F])	Incorrect temperature setting	Set correct temperature.
	Temperature controller malfunction	Calibrate or replace temperature controller.
	Dryer not designed to maintain	Repair or replace dryer.
	Thermocouple loose or malfunction	Repair or replace thermocouple.
	Heater malfunction	Repair or replace heater.

Injection molding

Proper conditions and machine operations for molding Eastman Tritan™ copolyester are discussed in this section.

Molding conditions

Barrel and melt temperatures

The first consideration in setting barrel temperatures is how much shot capacity will be used. Typically, if about half the machine’s shot capacity is used in each shot, barrel temperatures are set almost the same from back to front or slightly cooler at the feed end. If the shot is small relative to machine capacity, then temperatures are set significantly cooler at the feed end to minimize degradation due to long residence times at high temperatures. If the shot size is most of the machine’s capacity, flat or higher temperatures at the feed end are typically used. These polymers often require a descending profile with higher rear-zone set points to achieve proper screw recovery.

Another important factor is expected cycle time. For example, if the expected cycle time is long because of limited mold cooling, barrel temperatures should be lower. Different screws add different amounts of shear heat, but it is common to see melt temperatures 10°–20°C (20°–40°F) above the barrel settings.

Actual melt temperature should be checked with a needle pyrometer. Melt temperature is best taken when the cycle is established and an on-cycle shot is caught in an insulated container. (CAUTION: Care must be exercised when taking samples of HOT molten material.)

Melt temperature is the biggest factor in ease of filling the mold. Typically, melt temperatures 5°–20°C (10°–30°F) above the minimum temperature required to fill a part will give a good processing window. Melt temperatures on the high end tend to cause degradation and related problems.

Mold temperatures

Mold temperatures affect overall cycle time, shrinkage, warpage, and other characteristics of the molded part.

Eastman Tritan™ copolyester requires colder molds than some other plastics; therefore, anticipating cooling needs ahead of time (i.e., via tool design) pays dividends in reduced cycle time and processability.

High mold temperatures can cause sticking. Even localized hot spots where sticking might occur can extend the cycle.

Effective mold cooling requires:

- Ample mold-cooling channels with proper spacing and sizing
- Good cooling of pins, thin steel areas, and slides
- Good cooling near hot spots such as sprues or hot runners; insulating areas around hot runners
- Good water supply with few flow restrictions. Flow rate should be high enough to reach a minimum Reynolds number (N_r) of 6,000 in those areas where heat transfer (cooling) is desired. See the formula on the following page.

Flow rate can be increased and maintained by Thermolators®. Thermolators also reduce temperature variations that are common with central tower or chilled water systems.

Attention to these points will result in optimum cycle times with good surface appearance.

With good cooling, as previously outlined, the cooling phase of the cycle can be minimized so that the part is solidified and easily ejected.

Larger-diameter sprues may still be soft and rubbery. Additional cooling could be needed to prevent sprue sticking. Review “Mold design and construction” for additional information.

Reynolds number formula

N_r less than 2,000 is laminar.
N_r 2,000 to 3,500 is in transition.
N_r greater than 3,500 is turbulent flow.

Metric

N_r = 1,000 VD/n or 3,160 Q/Dn

V = fluid velocity in m/sec
D = diameter of passage in mm
n = kinematic viscosity in centistokes (see Table 6)
Q = coolant flow rate in liters/min

U.S. customary

N_r = 7,740 VD/n or 3,160 Q/Dn

V = fluid velocity in ft/sec
D = diameter of passage in in.
n = kinematic viscosity in centistokes (see Table 6)
Q = coolant flow rate in gal/min

Table 6 Kinematic viscosity for water

°C	°F	Viscosity centistokes (mm²/sec)
0	32	1.79
4	40	1.54
10	50	1.31
16	60	1.12
21	70	0.98
27	80	0.86
32	90	0.76
38	100	0.69
49	120	0.56
60	140	0.47
71	160	0.40
82	180	0.35
93	200	0.31
100	212	0.28

Injection speed

To minimize gate blush, splay, or both, the fill speed used for Eastman Tritan™ copolyester is slower than for some other plastics. Machines with fill-speed programming capability are recommended. Start the fill at a very slow speed, such as 10%–20% of available capacity for the first 3%–15% of the shot, then increase to 40%–60% to complete the shot. An average fill rate of 50–250 g/sec (1.76–8.8 oz/sec) is typical.

Screw speed

The screw should be run at the minimum rpm that will allow it to recover 2–5 seconds before the mold opens. This minimizes viscous heat generation, tends to make the melt more uniform, and minimizes dead time.

Pack and hold

A common problem with direct sprue-gated parts is a shrinkage void at the base of the sprue. Long hold times of 8–12 seconds and lower hold pressures of 275–550 bar (4,000–8,000 psi) (nozzle plastic pressure) will feed material to the sprue at a rate that will eliminate voids but not overpack the sprue. Overall cycle time does not have to be extended if the cooling timer is decreased by the amount the hold timer is raised. A shrinkage void can also form with a conventional runner at the junction of the runner and sucker pin; this can also be eliminated by using the same methodology.

Cushion size

Cushion size should be at the absolute minimum to ensure the screw does not hit bottom and the pack-and-hold pressures are transmitted to the part. The cushion left at the end of the pack-and-hold phase of the cycle is typically 3–13 mm (0.125–0.5 in.), depending on machine size and injection speed. Larger cushions can increase holdup time in the barrel and contribute to degradation.

Continued forward movement of the screw at the end of the shot indicates a leaking check valve. A leaking check valve will prevent a cushion from being maintained and can cause random short shots and shot-to-shot variability.

Back pressure

Typical back pressure is 7–10 bar (100–150 psi), though it may be as low as 3.5 bar (50 psi). To improve melt uniformity, increase melt temperature, or eliminate air entrapment (air splay), back pressure can be increased to as much as 28 bar (400 psi). Excessively high back pressures can aggravate drooling into the mold, since decompression is usually kept to a minimum.

Decompression

In general, minimal decompression is used. Decompression tends to pull air back into the nozzle, causing splay in the next shot. Small amounts of decompression can be used to reduce drool.

Trial preparation and operation

Before beginning a trial, be sure that all conditions are optimized.

- Check the dryer for proper:
  - Air temperature at the hopper entrance
  - Dew point of the drying air
  - Airflow
  - Hopper capacity relative to the size of sample to be dried
- Clean the hopper and material-handling system thoroughly.
- Ensure that proper mold-temperature control is available.
- Check the chiller and Thermolator®.
- Be sure the molding machine is clean. (See the next section on start-up and purging.)
- Determine trial objectives. The purpose of the trial dictates the amount of material needed and the quality and number of parts required. For example, if the purpose is to obtain 5 good parts for testing, it is typical to run 200 shots on an untried mold. If the purpose is to make 50 parts, minimize cycle, or check part variability, the amount of material and time required will be much greater.

Form 1 (Pretrial preparation) and Form 2 (Molding conditions record) can be used during the molding trial. Completing Form 1 helps ensure that all needed preliminary operations have been performed. Changes made to processing variables and effects of those changes during the trial or start-up can be documented on Form 2.

- Retain 20–60-g (0.70–2.11-oz) samples of the pellets and parts for follow-up testing of IV or molecular weight.
- Document and save all setup conditions, changes to conditions, and their effects on part quality. Add comments regarding what worked well and what caused problems. Provide copies to all trial team members and to your Eastman representative. When the job goes to production, give copies to all persons involved.

Start-up

Start with a clean machine. If the machine is not purged, unmelted particles, gaseous splay, or a combination of problems will result. Ball checks are typically slow to purge and generally are not recommended; check rings are preferred.

Removing and cleaning the screw, check valve, nozzle, and barrel are the only effective means of purging difficult-to-remove high-temperature plastics.

Purge materials. The material most effective in purging is a polymer similar to the material to be run.

Polyethylene and polypropylene should be avoided because they can mix with the new material and cause streaks for extended periods of time. For difficult-to-remove materials, nozzle and front-barrel-zone set points are sometimes increased up to 300°C (570°F) to soak and purge then cooled back to running temperatures. Use caution and refer to the manufacturer's recommendations for the material used in the previous run.

After any cycle interruption of more than approximately 5 minutes, purging 3 to 5 shots is good practice.

Form 1 Pretrial preparation

Company ____________________ Date sch. ______________ Material ____________________

Location ____________________ Time sch. ______________ Pounds ____________________

Part ____________________

Received? ____________________

Trial objectives (short demo, fast cycle, production run, etc.) ______________________________

Technical contact/phone ____

Dryer

Desiccant dryer ________, capacity ____________ lb, temp __________ °C/°F, time __________

Part and shot (Part available to send in will eliminate several of the following items.)

Part weight ____________

Specific gravity ____________

Flow length from gate ____________

Wall thickness(es) ____________

Gate size ____________

Number of cavities ____________

Shot weight ____________ ounces

Part projected area ____________ sq inches

Molding machine

Molding machine capacity ____________ ounces

Machine clamp ____________ tons

Inject speed profile ability? ____________

Mold cooling available? ____________

Check valve and screw type ____

Hot-runner system/type ____

Other information

Estimated cycle time ____________ seconds

Current production material ____________

Annual volume ____________

Previous trials ____

Anticipated/typical problems ____

Lit./info. to customer on processing? ____

Starting barrel and mold temp ____

Purge (typically acrylic or polystyrene, not polypropylene or polyethylene) ____

Form 2 Molding conditions record

Company ____________________
Date/time ____________________
Part ____________________
Mold No. ____________________
No. cavities ____________________
Machine No.____ size____ tons____ ounces________
Screw type ____________________
Check-ring type ____________________
Nozzle orifice (size) _______ Hot runner ____________

Company representative ____________________
____
____
____

Eastman representative ____________________
____
____
____
____

	Run No.					
	Material—color					
	Lot No.					
Drying	Dew point					
	Temperature					
	Time (hours)					
Temperatures	Feed zone					
	Center zone					
	Front zone					
	Nozzle					
	Hot manifold					
	Actual melt					
	Mold—fixed					
	Mold—moveable					
	Mold—slides					
Pressure	Clamp					
	Filling pressure					
	Packing pressure					
	Hold pressure					
	Back pressure					

Form 2 Molding conditions record *(continued)*

Cycle times	Injection, total screw forward					
	Fill time					
	Pack time					
	Hold time					
	Cooling					
	Plasticizing					
	Total open					
	Overall					
	Residence time					
Miscellaneous	Transfer method					
	Transfer position					
	Transfer weight (%)					
	Decompression (length)					
	Cushion					
	Screw rpm					
	Total shot (weight)					
	Single part (weight)					
Comments	A—					
	B—					
	C—					
	D—					
	E—					

Production molding

Production start-up

The processing window needs to be defined to establish a controlled molding process. The starting point for a production run is typically the same as the conditions used in the last molding trial or start-up run. The machine should be cleaned before production is started. After the window is defined, routine production operations should be set in the middle of the window so that normal variability does not result in scrap parts.

Cycle uniformity or rhythm. To maintain shot-to-shot consistency, it is best to maintain a constant cycle. With manual part removal (semiautomatic operation), a good rhythm should be established to maintain a constant time in the barrel from shot to shot.

Scrap minimization. To minimize scrap, first determine the sources of scrap. Next, attempt to correct the largest sources first. Form 3 is provided as a tool for defining the problems that are causing scrap. After the cause is determined, refer to the “Troubleshooting guide” for suggested solutions to various problems.

Use of regrind

It is generally suggested that the regrind feeding rate be kept to 20% or less. This will help maximize part quality.

The quality of the regrind is as important as the quality of the virgin material.

Regrind should be:

- Kept free of contamination
- Ground with sharp grinders to minimize fines and overheating
- Dried
- Fed at a constant rate

Some typical sources of contamination include:

- Hot stamping
- Heat transfer tape
- Contaminated parts
- Purging

Shutdown

In general, the feed can be shut off and molding continued on cycle until the screw is run dry. If you are changing to another material, purge with a polymer similar to the material to be run. Run the screw dry, and turn off the power.

Always leave the screw forward; otherwise, a large slug of material must be remelted. If the slug does not fully melt before the screw is injected forward, check-ring damage may result.

Packaging and part handling

Parts need to be protected from being scratched, dented, or otherwise damaged during handling. Packaging should be planned to protect the finish that Eastman copolyesters can produce.

Form 3 Form for identifying problems that are causing scrap

Title ______________________________
Company __________________________
Machine No. _______________________

Name ______________________________ Part name ______________________________
Date ______________ Shift 1 2 3 4 5 Part No. _______________________________

Scrap reason	2	4	6	8	10	12	14	16	18	20	22	24	26	28	30	32	34	36	38	40	42	44	46	48	50	Total	% of total
																										Total scrap		

Total good ☐ Total scrap ☐ Total run ☐

% total scrap ________

Troubleshooting guide

Molding Eastman Tritan™ copolyester

Suggested remedies. **Do 1 first, 2, 3, etc.** **+ means increase** **− means decrease**	Problems																	
	Short shots	Brittle parts	Voids or sinks[a]	Bubbles[a]	Surface splay	Gate splay	Sprue sticking	Part sticking	Burning	Flash	Brown streaks	Discoloration	Black specks	Weld lines	Jetting	Warpage	Fine waves	Large irregular waves
Drying																		
Make sure resin is dry.		1	11		1	5	12	7		10		4−						
Temperatures																		
Melt temperature	6+	2−	7+		4±	6±	13−			7−		3−	6±	1+	3+		3+	
Mold temperature	7+		9±					3−		8−				7+		1±		3−
Nozzle or hot runner	8+				5−						2−	6−			4+	4+		
Decrease sprue temperature							5											
Reduce ΔT (barrel, nozzle, HR)					6													
Pressures																		
Injection pressure	2+	10−	3+			4−	8−	4−		1−				3+	2±	7−	2+	1+
Pack pressure																		2+
Back pressure	9+	6−	8+	1+	7±		11+					8−	4±					
Clamp pressure										3+								
Times																		
Mold closed/cooling			4−				6−	9±								2±		
Injection hold/pack	5+		2+				9−	5−		4−				5+		8		4+
Booster time	3+		5+				10−	6−		2−				4+		9		5+
Reduce overall cycle		4−			12−			10				9−						
Other																		
Injection speed	4+		6−		2−	1−	4−		2−	5−				6+	1±	3−	1+	6±
Screw speed		5−	10+	4±	10−							7−	5					
Decompression/ suck back				2−	3−	3−			5									
Use sprue break							7											
Cushion size	1+		1+	3														
Ejection uniformity								8								5		
Purge barrel[b]				6	8				3		3	1	1					
Reduce regrind %		7	12	8				11		9		5	3					
Check for voids		8																
Check for contamination		3		7	9				4			2	2					
Hardware																		
Eliminate sharp corners					14													
Check sprue/ nozzle diameter				5			1				4	10						
Increase gate size/ streamline	10+		13+		11+	7+									5+			
Change gate location			14	9					6	12				9	6	11		
Vent size/ cleaning/location	11+								1+	6−				2+				
Add overflow @ weld area														8				
Use different size machine	12+	9−			13					13+		11						
Repair mold	13			10			15	13		11								
Increase taper/ draft angle							14	12								10		
Check for undercuts							3	1−								6−		
Check puller design							2	2										
Fix defective check ring											1	12	7					
Add pin, etc., to impinge flow															7			

General guidelines:

- Dry at −30°C (−20°F) or lower dew point.
- Dry for 6 hours.
- Use 0.06 m³/min dry air per kg/h (1 ft³/min dry air per lb/h) of resin used.
- Refer to Data Sheet for correct drying temperature.
- Use multidesiccant bed dryer (vs. tray or batch dryer).
- Select machine size to use 40%–75% of capacity.
- Use sliding check ring with no holdup spots rather than ball check.
- Use full round runners.
- Use nozzle diameter greater than 4.8 mm (3⁄16 in.).
- Use short nozzle or nozzle with uniform heat control.
- Use moderate back pressure, usually 0.34–1.38 MPa (50–200 psi).
- Use slow to moderate screw speeds.
- Gate into thickest area of part with gate 50%–75% of part thickness.
- Vent mold 0.0245–0.0305 mm (0.001–0.0012 in.).

[a] Voids typically have no foamy tail where air/gas bubbles often have a tail or trail.

[b] Purge with commercial purge compounds, acrylic, styrene; polyethylene and polypropylene are not usually good purge material for these resins.

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

Safety Data Sheets providing safety precautions that should be observed when handling and storing Eastman products are available online or by request. You should obtain and review the available material safety information before handling any of these products. If any materials mentioned are not Eastman products, appropriate industrial hygiene and other safety precautions recommended by their manufacturers should be observed.

It is the responsibility of the medical device manufacturer ("Manufacturer") to determine the suitability of all component parts and raw materials, including any Eastman product, used in its final product to ensure safety and compliance with requirements of the United States Food and Drug Administration (FDA) or other international regulatory agencies.

Eastman products have not been designed for nor are they promoted for end uses that would be categorized either by the United States FDA or by the International Standards Organization (ISO) as implant devices. Eastman products are not intended for use in the following applications: (1) in any bodily implant applications for greater than 30 days, based on FDA-Modified ISO-10993, Part 1, "Biological Evaluation of Medical Devices" tests (including any cosmetic, reconstructive, or reproductive implant applications); (2) in any cardiac prosthetic device application, regardless of the length of time involved, including, without limitation, pacemaker leads and devices, artificial hearts, heart valves, intra-aortic balloons and control systems, and ventricular bypass assisted devices; or (3) as any critical component in any medical device that supports or sustains human life.

For manufacturers of medical devices, biological evaluation of medical devices is performed to determine the potential toxicity resulting from contact of the component materials of the device with the body. The ranges of tests under FDA-Modified ISO-10993, Part 1, "Biological Evaluation of Medical Devices" include cytotoxicity, sensitization, irritation or intracutaneous reactivity, systemic toxicity (acute), subchronic toxicity (subacute), implantation, and hemocompatibility. For Eastman products offered for the medical market, limited testing information is available on request. The Manufacturer of the medical device is responsible for the biological evaluation of the finished medical device.

The suitability of an Eastman product in a given end-use environment is dependent on various conditions including, without limitation, chemical compatibility, temperature, part design, sterilization method, residual stresses, and external loads. It is the responsibility of the Manufacturer to evaluate its final product under actual end-use requirements and to adequately advise and warn purchasers and users thereof.