

Eastman Tritan™ copolyester— Design 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 clears 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.

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 and 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 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)

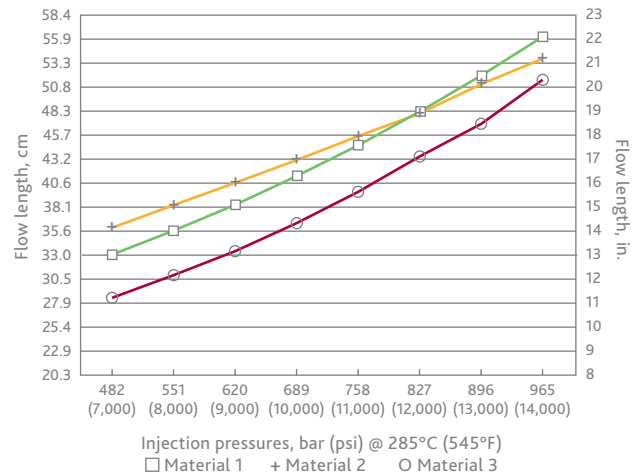
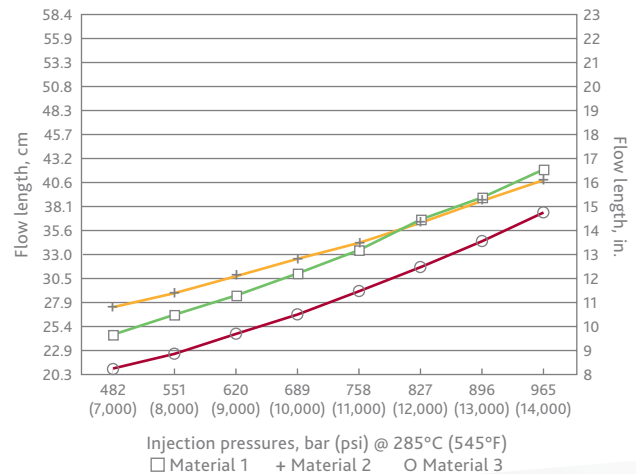


Figure 2. Spiral flow data at midrange temperatures (2.8-mm [0.110-in.] wall)



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	
Mechanical	Clamp tonnage prediction
Thermal	Flow front temperature
Viscosity	Time required to freeze
Pressure, volume, temperature data	Volumetric shrinkage
Processing conditions	
Mold temperature	100°–140°F (38°–60°C)
Melt temperature	540° ± 20°F (282° ± 10°C)
Injection profile	3 mm (0.5 in.)/g first 5%–15%, 43 mm (1.7 in.)/g, then 23 mm (0.9 in.)/g
Packing profile	8–12 seconds @ 34–52 MPa (5,000–7,500 psi)

Mold filling analysis

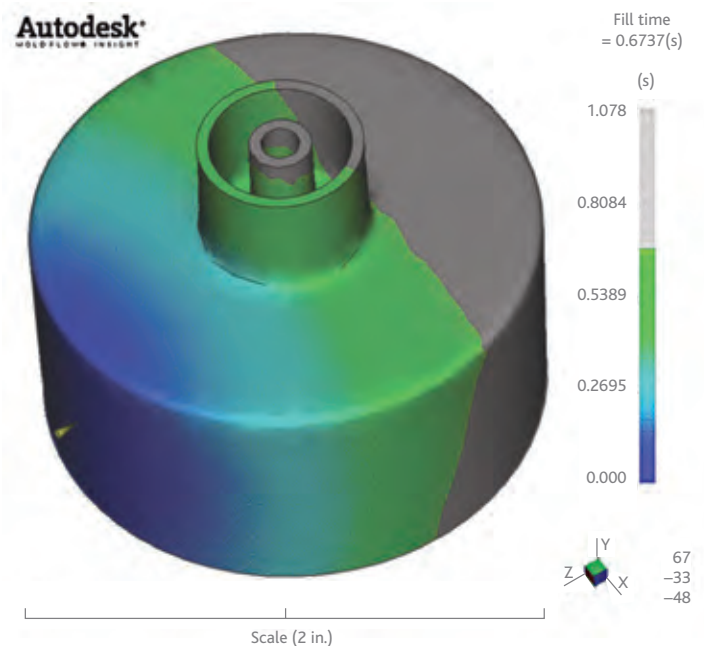
Computer-aided mold filling analysis is particularly useful in part and tooling design. Flow patterns can be observed to determine the existence of flow imbalances, which can be corrected by adjusting wall thickness, placement of flow leaders, or both. Imbalanced fill can result in underpacked areas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

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

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.

Figure 3. Computer-aided mold filling analysis image



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

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™ MX711 copolyester are shown in Figure 4.

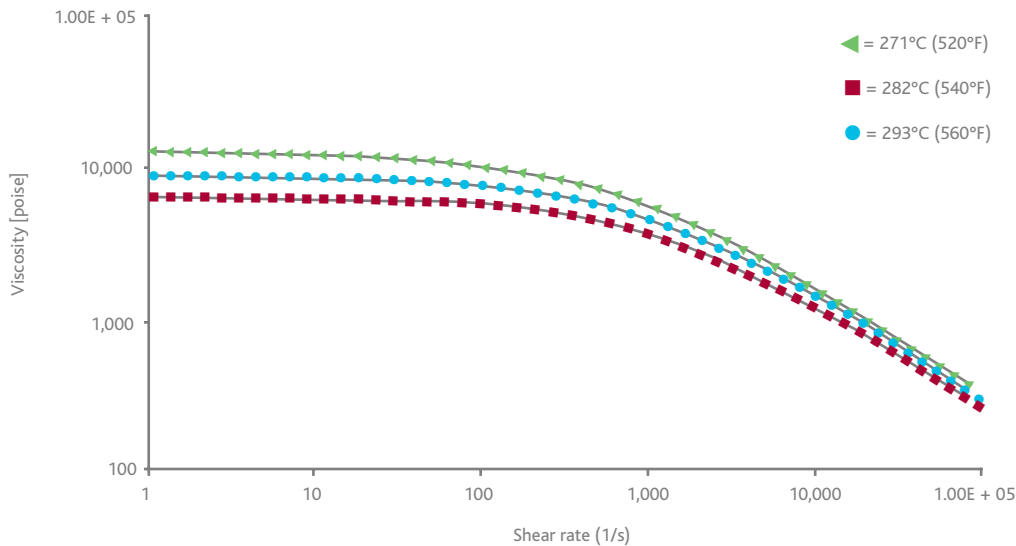
Molten plastics typically exhibit non-Newtonian flow behavior; that is, their resistance to flow does not vary linearly with the pressure applied. The material is said to be shear-thinning if doubling the pressure results in more than twice the flow rate. The viscosity of Eastman polyester plastics has 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.

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 one-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 page 6.

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



Mold design

Tool steels and mold construction

There are several factors to consider when selecting steel for the mold—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 like 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 requiring higher thermal conductivity may be constructed with alloys such as MoldMax HH, which has a hardness of 40 Rc, or Moldstar 90, which has a hardness of 30 Rc.

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 production.

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 (⅜–⅝ 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 the following:

- **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 to 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.

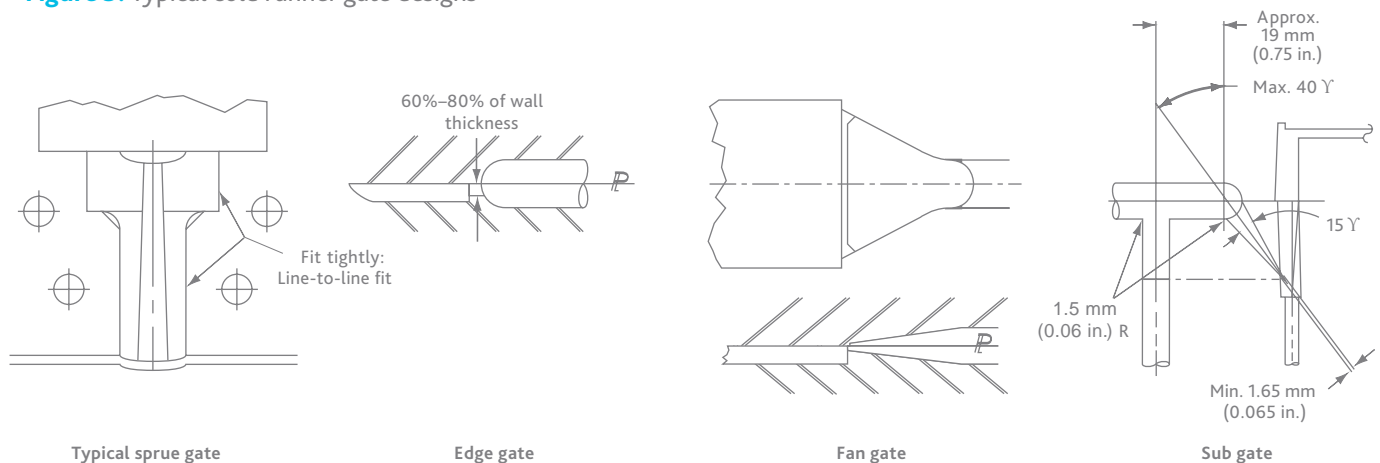
The hot probe-style drop needs to have its own thermocouple, heat source, and control to allow regulation of the temperature as needed. The thermocouple should be located near the probe tip for accurate temperature control with minimal variation.

- **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.
- **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.
- **Minimizing shear heating.** The diameter of the flow path needs to be large enough to minimize 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.

Figure 5. Typical cold runner gate designs



Sprue design

Proper sprue design is important for good molding and for 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. Enough 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.

Use of thermal sprue bushings made from a high heat transfer tooling material such as is strongly suggested. Performance sprue bushings in standard sizes constructed with Moldstar 150 are available from Performance Alloys at <http://www.moldstar.com/spruecpec.html>. 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 to remove any undercuts. If sprue sticking still occurs, then a bushing having 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.

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 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 will also permit reduction of cycle time.

Figure 6. Drilled coolant lines

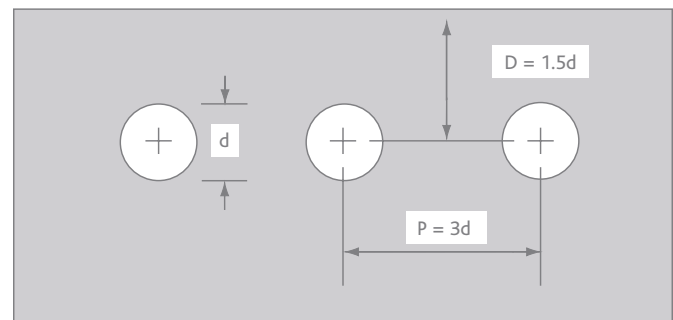
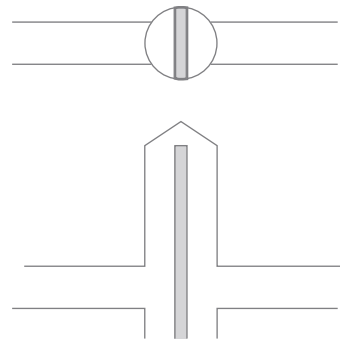


Figure 7. Baffled coolant drops



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

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.

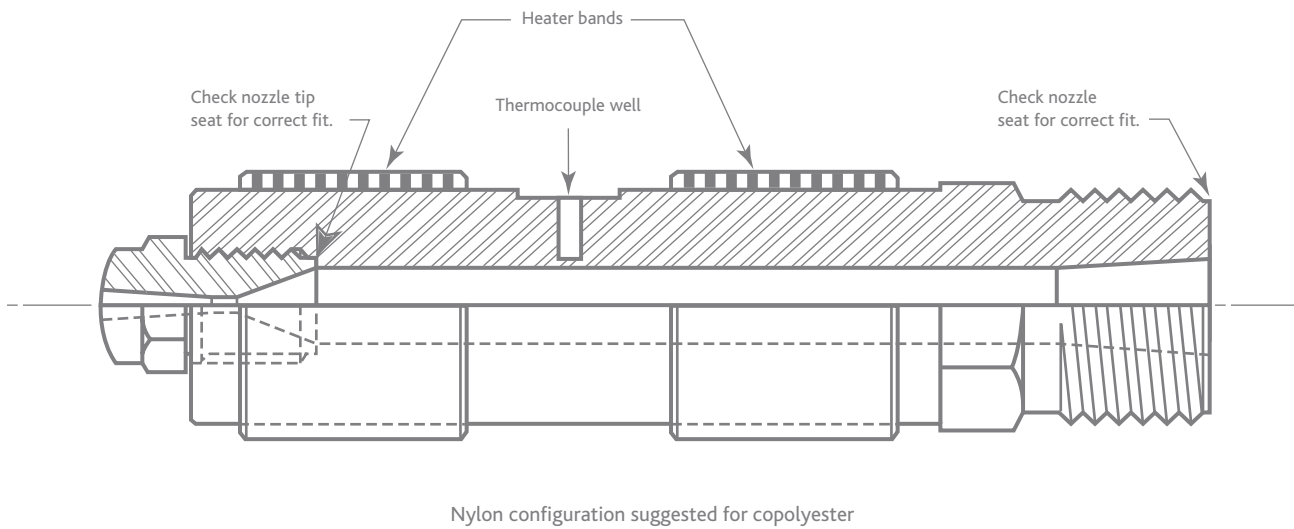
Cold runner mold construction guidelines for Eastman Tritan™ copolyester

- 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 ($\frac{5}{32}$ – $\frac{9}{32}$ in.) diameter. Larger parts will need orifice diameters of 7 mm ($\frac{9}{32}$ in.) while smaller parts will need only 4 mm ($\frac{5}{32}$ 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 ($\frac{7}{32}$ 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 possess significantly better thermal efficiency than traditional steel sprue bushings.

Hot runner mold construction guidelines for Tritan

- 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.

Figure 8. Injection nozzle



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 ($\frac{3}{16}$ – $\frac{5}{16}$ in.) are typical for smaller parts; those for larger parts should have a 9.5 mm ($\frac{3}{8}$ 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 setpoint of the controller or, preferably, to 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 8.

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 to 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.

General processing guidelines for design consideration

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, then increasing to 43 mm (1.7 in.) per second, 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 non-streamlined 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 steadier and more 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 tends 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 to assure 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.

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.

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

Tritan 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 18:1–22:1. 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.

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



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

Although the information and recommendations set forth herein are presented in good faith, Eastman Chemical Company ("Eastman") and its subsidiaries make no representations or warranties as to the completeness or accuracy thereof. You must make your own determination of its suitability and completeness for your own use, for the protection of the environment, and for the health and safety of your employees and purchasers of your products. Nothing contained herein is to be construed as a recommendation to use any product, process, equipment, or formulation in conflict with any patent, and we make no representations or warranties, express or implied, that the use thereof will not infringe any patent. NO REPRESENTATIONS OR WARRANTIES, EITHER EXPRESS OR IMPLIED, OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR OF ANY OTHER NATURE ARE MADE HEREUNDER WITH RESPECT TO INFORMATION OR THE PRODUCT TO WHICH INFORMATION REFERS AND NOTHING HEREIN WAIVES ANY OF THE SELLER'S CONDITIONS OF SALE.

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

© 2020 Eastman. Eastman brands referenced herein are trademarks of Eastman or one of its subsidiaries or are being used under license. The ® symbol denotes registered trademark status in the U.S.; marks may also be registered internationally. Non-Eastman brands referenced herein are trademarks of their respective owners.