Processing guidelines for DuraStar™ DS2000/DS2010 polymer

Introduction

DuraStar™ polymers afford the consumer brilliantly clear polymers as well as excellent impact strength, chemical resistance, dimensional stability, low shrinkage rates, and other enhanced physical property advantages. These processing guidelines are given to help optimize these physical properties and widen the processing window.

Drying

- Use a desiccant-type drying system with dry air at a minimum dew point of –29°C (–20°F).
- Dry DuraStar polymer at 70°–75°C (158°–167°F) for 2 hours minimum. Residence time in the dryers can be up to 24 hours. The inlet air temperature needs to be controlled within ± 3°C (± 5°F) throughout the drying cycle.
- The dryer should have sufficient airflow to assure a uniform pellet temperature throughout the dryer. A minimum of 3.7 m^3/h airflow is suggested for each kilogram of polymer processed per hour (1.0 cfm per pound per hour of polymer processed).

Molding machine

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

Processing

Barrel and melt temperatures

- Processing at the lowest possible recommended processing temperature and minimum residence time in the machine will maximize physical properties.
- Well-dried material is key to shot-to-shot conformity. Engineering materials tend to be more sensitive to degradation at their processing temperatures due in part to the hygroscopic nature of these polymers.
- Normal processing temperatures are in the range of 240° to 265°C (465° to 510°F) air shot. When molding parts at faster cycle times utilizing larger barrel capacity, such as 50%–80%, the typically resulting lower residence generally allows running at the higher range of the melt temperature without encountering significant material degradation. Contrary to this, when parts are molded with long cycle times utilizing a minor amount of the barrel capacity, such as 10%–25% even though it should be avoided when possible, the processor should strive to run the polymer at the lower range of the proposed melt temperature.
- A flat temperature profile setting is normally used; i.e., a barrel with a three-zone system might have settings as follows:

  Rear zone: 240°C (465°F)
  Center zone: 240°C (465°F)
  Front zone: 240°C (465°F)
  Nozzle: 240°C (465°F)

- 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 to account for additional temperature increase due to shear heating. It is good practice to determine the actual melt temperature using a pyrometer. Also, it is important that the casting around the throat of the injection molding machine be cooled to provide optimum pickup of the material.

Mold temperature

Mold temperatures ranging from 13° to 30°C (55° to 84°F) produce the best parts.

- DuraStar polymers require 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, 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, the cooling part 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 for additional information.

Fill speed

- Fill speeds used for DuraStar™ polymers are slower than typical plastics. Machines with fill-speed profile capability are recommended. Where fill-speed profiling is available on a machine, starting the fill at a very slow speed such as 13 mm (0.5 in.) per second for the first 5% to 15% of the shot, then 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. Where direct sprue gating into the part is used, a moderate to fast fill rate such as 38 to 56 mm (1.5 to 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, radiuses may need to be added to these features to reduce blush near the gate. Gate thickness, as well as injection speed, can influence gate blush. Gate thicknesses of less than 1.1 mm (0.045 in.) are not suggested for most gate types.

Screw speed (rpm)

- The rpm of the screw should be slowed to the minimum speed necessary to recover the screw during part cooling and should sit at the rear position for only 2 to 5 seconds before the mold opens. This minimizes high-speed shear and tends to make the melt more uniform.

Pack and hold

- Where direct sprue gating into the part is used, longer hold times in combination with lower hold pressures might 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 so that it will release with the part. Having long hold times of 8 to 12 seconds and lower hold pressures of 34 to 52 MPa (5,000 to 7,500 psi) will feed material to the sprue to fill the void while not over packing the sprue. The overall cycle time would not be affected if the cooling time proportionally decreased by the length of additional holding time. Sticking can also occur 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 to eliminate the problem.

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 getting into the part. The cushion left at the end of the pack and hold is typically 5 to 10 mm (0.2 to 0.4 in.) depending on machine size and injection speed. Larger cushions can add to hold time in the barrel and aggravate degradation. If the screw continues to move forward at the end of the shot after adequate time is given to come to a stop, it is a sign of a leaking check valve, which may also cause short shots and shot-to-shot variability.

Back pressure

- Back pressure is usually minimum at about 10 MPa (1,500 psi).[1] To improve melt uniformity (and mix concentrates), increase melt temperature, or to get rid of air entrapment (air splay), back pressure can be increased gradually to as much as 15.5 MPa (2,250 psi).[1] Higher back pressures can aggravate drooling into the mold.

Decompression (suck back)

- In general, use very small or no decompression because it 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

- General-purpose screws with compression ratios in the 2.8:1 or 3:1 range and L/D ratios of 20:1 have been used successfully. 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. These unfilled materials are generally very mild on screw wear. Corrosion of barrel and screw parts is not expected with DuraStar™ polymers.

Purging

Purging from other polymers to DuraStar DS2000/DS2010:

- Purging can be carried out with clear undried polycarbonate or clear polycarbonate regrind at melt temperatures of 270°–290°C (518°–555°F) to eliminate the previous polymer. For applications where there may be concerns with using polycarbonate, e.g., in order to avoid BPA in molded products, purging directly with DuraStar is recommended. After an adequate amount of purging, which will vary depending on the previous polymer molded, the barrel of the injection molding machine can be followed directly with DuraStar polymer without further purging.

Purging from DuraStar DS2000/DS2010 to other polymers:

- Purge with acrylics, PS, commercial purging compounds, or the polymer that follows the DuraStar™ polymer.

[1]Note that this is actual hydraulic pressure, not molding machine gauge pressure.

Mold construction guidelines for DuraStar™ polymers

These guidelines can be used to minimize sprue sticking, reduce cycle time, and widen the processing window.

- Taper to 3° minimum (included angle) on the sprue bushing.
- Orifice size of the sprue bushing where it meets the nozzle should be 4 to 7 mm (5⁄32 to 9⁄32 in.) diameter. Larger parts will need orifice diameters of 7 mm while smaller parts will need only 4 mm.
- Shorten the sprue bushing to less than 75 mm (3 in.) in length.
- The sprue bushing should have a high polish in the sprue area.
- Increased cooling around the sprue bushing—upper and lower water line circuits are recommended.
- Good surface contact between the sprue bushing and mold surface.
  - Line-on-line interference fit is recommended.
  - Surface contact should be on the head of the sprue bushing as well as the shaft.
- Water line spacing 50 to 64 mm (2 to 2.5 in.) between center lines.
- Air poppets should be offset as far as possible from the center line of the sprue.

For example, a sprue bushing for a medium size 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.).

In cases where aggressive molding cycles are desired, substitute a Performance Products 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.

EASTMAN

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

Telephone:
U.S.A. and Canada, 800-EASTMAN (800-327-8626)
Other Locations, (1) 423-229-2000
Fax: (1) 423-229-1193

Eastman Chemical Latin America
9155 South Dadeland Blvd.
Suite 1116
Miami, FL 33156 U.S.A.

Telephone: (1) 305-671-2800
Fax: (1) 305-671-2805

Eastman Chemical B.V.
Fascinatio Boulevard 602-614
2909 VA Capelle aan den IJssel
The Netherlands

Telephone: (31) 10 2402 111
Fax: (31) 10 2402 100

Eastman (Shanghai) Chemical
Commercial Company, Ltd. Jingan Branch
1206, CITIC Square
No. 1168 Nanjing Road (W)
Shanghai 200041, P.R. China

Telephone: (86) 21 6120-8700
Fax: (86) 21 5213-5255

Eastman Chemical Japan Ltd.
MetLife Aoyama Building 5F
2-11-16 Minami Aoyama
Minato-ku, Tokyo 107-0062 Japan

Telephone: (81) 3-3475-9510
Fax: (81) 3-3475-9515

Eastman Chemical Asia Pacific Pte. Ltd.
#05-04 Winsland House
3 Killiney Road
Singapore 239519

Telephone: (65) 6831-3100
Fax: (65) 6732-4930

www.eastman.com

PP-20B 9/13