Santoprene™ TPVs

ExxonMobil

Processing guidelines for injection molding of thermoplastic vulcanizates

Energy lives here™

INTRODUCTION AND SAFETY GUIDELINES 5

EQUIPMENT 6

GENERAL PROCESSING

- Start up 8
- Mold filling, packing and screw rotation 8
- Estimating molding cycle time 9
- Drying of the material 10
- Coloring 10
- Regrinding 11
- Shrinkage 11

MOLD DESIGN

- Mold steel selection 12
- Mold surfaces 12
- Mold plating and coating 13
- Mold venting 13
- Sprue, runner and gate design 14
- Ejection 19
- Undercut design guidelines 19
- Mold temperature control 19

HOT RUNNER SYSTEMS

- System selection 20
- Mold design 20
- Gate selection 21

MULTI-MATERIAL MOLDING

- Overmolding 23
- Insert-molding 23
- Two-shot molding 23
- Co-injection molding 23

QUICK REFERENCE NOTES 24

TROUBLESHOOTING 24

INTRODUCTION AND SAFETY GUIDELINES

Santoprene™ thermoplastic vulcanizates (TPVs) from ExxonMobil Chemical combine the processability of thermo plastic materials with many of the performance characteristics of thermoset rubber. Santoprene TPVs are commonly employed in the plastics industry for efficient and easy processing with conventional injection molding equipment. They are suitable for a broad range of systems that require excellent part definition.

TPVs are categorized as a type of thermoplastic elastomer (TPE). They contain very small fully vulcanized rubber particles dispersed in a plastic matrix. This fully vulcanized structure gives a rheology similar to a composite and a flow behavior that is pseudoplastic. Typically TPEs and TPVs have a melt viscosity that is more dependent on shear than on melt temperature. To make handling easier, we supply our Santoprene TPVs in pellet form. Most TPEs and TPVs absorb moisture, and predrying is recommended (see Drying section for more information.)

The processing guidelines included in this document are intended only for general purpose and molding grades of Santoprene TPVs from ExxonMobil Chemical. Other specialty TPV grades may require different processing procedures.

Safety guidelines

ExxonMobil Chemical is committed to continually improving the safety of operations so they are free from injury or incident. Remember, the typical processing operation uses powerful and potentially dangerous electrical, hydraulic and mechanical systems. Material temperatures can be as high as 290 °C (550 °F). Hot and cold water and hot machine oil can be encountered. Industry best practices strongly recommend the following when processing TPVs:

- Check all safety systems (electrical, hydraulic and mechanical) at least once daily to ensure they are fully operational.
- Fix all leaks on or around mac ; avoid oil and water spills on the floor.
- Purge all previous materials from the press using an LDPE or PP (some of our materials are not compatible with other thermoplastic materials please refer to each material's MSDS and product data sheet prior to use).
- Wear the proper personal protective equipment while in any processing area.
- Follow all the safety recommendations from the machinery and material manufacturers.

For more information, contact the AnswerPerson℠ at:
tpe.answerperson@exxonmobil.com or visit us at
www.santoprene.com

Figure 1
Exterior view of a typical thermoplastic injection molding machine

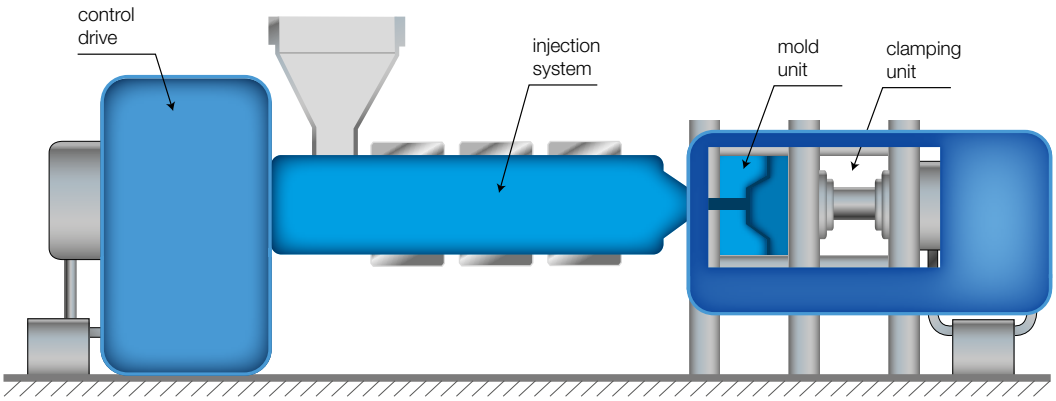
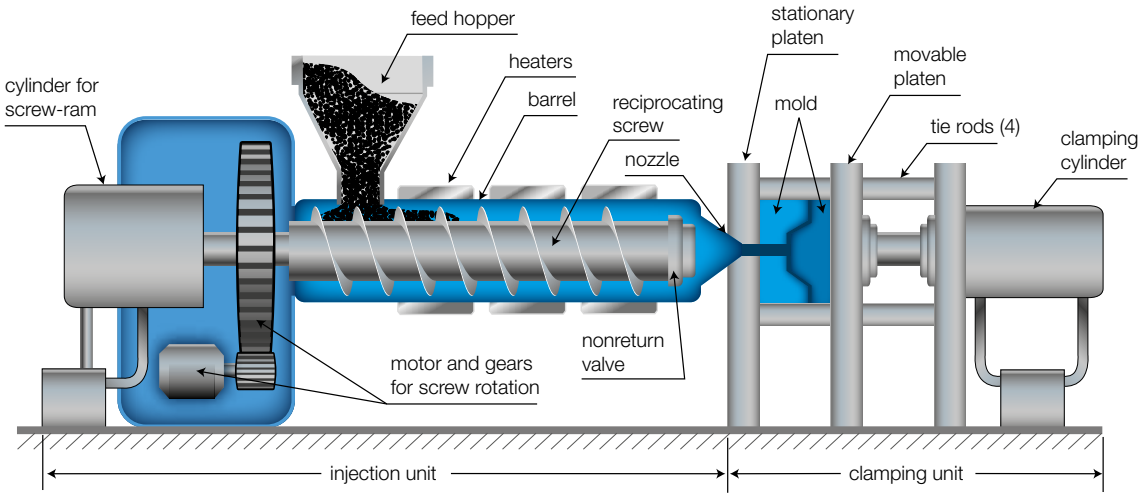


Figure 2
Cross section of a typical thermoplastic injection molding machine



EQUIPMENT

Machine selection

Santoprene TPVs from ExxonMobil Chemical have been processed successfully on many makes and sizes of standard reciprocating screw injection molding machines. The quality of a final molded part depends on the ability of a machine to deliver the optimum amount of plasticized material for the intended application.

Screw design

In most cases, a generalpurpose polyolefinic screw with a compression ratio of 2:1 to 2.5:1 and a length to diameter ratio between 16:1 and 22:1 is sufficient. A low shear screw, such as those used in PVC processing, is not recommended. Unlike many thermoplastic materials, Santoprene TPVs demonstrate better flow characteristics during the injection filling phase when subjected to high levels of shear during fill.

Barrel capacity

Avoid underutilization of the barrel wherever possible, since it can lead to long residence times. Small shots run on a large capacity barrel complicate processing. The best practice for any injection molding is to utilize 20 to 80% of the barrel capacity for each shot. This typically translates to 1.3 to 5 shots in the barrel.

Nozzle

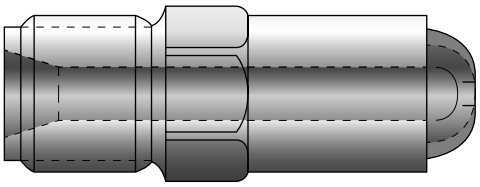
Most common TPVs do not typically drool or degrade at normal processing temperatures and therefore do not require a special nozzle. A conventional generalpurpose nozzle is commonly suited to most applications (see Figure 3).
The nozzle orifice is usually 10 to 15% smaller than the "O" diameter of the sprue bushing. Hot runner manufacturers recommend that hot sprues should run at the same temperature as the nozzle setting or slightly higher.

Recommended clamping force

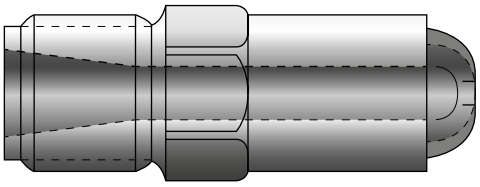
Follow typical TPV recommendations for clamping force of 4.0 to 6.9 kN/cm^2 (3 to 5 tons/in^2) of the projected area.

Figure 3
Nozzle types

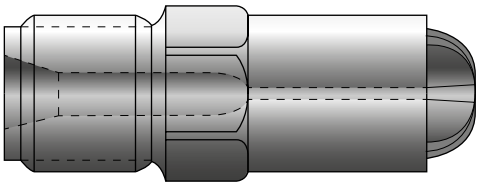
General purpose taper



Full internal taper



Nylon reverse taper



GENERAL PROCESSING

Start-up

General conditions

Most of our TPVs process with mediumfast to fast injection rates. Fill times are typically between 0.5 and 2.0 seconds. The melt temperatures for most materials are approximately 205 °C (400 °F); injection pressures are moderate. Starting conditions can be found in the Product Data Sheet for each grade and/or the Quick Processing Reference. These contain detailed information on processing setup and unique features. Both documents are available on **www.santoprene.com**

Parts made with Santoprene TPVs ordinarily release freely from the mold when following general industry guidelines for part and mold design and processing recommendations. The use of mold release sprays or powders is not recommended, except under special circumstances.

Injection molding pre-start-up

1. Purge barrel using a generalpurpose polypropylene or other suitable purging compound.
2. Set heat zones to obtain melt temperature near the middle of the recommended range for the TPV being used.
3. Set mold temperature to approximately 20 to 30 °C (70 to 80 °F).
4. Set screw speed to 100 to 200 rpm.
5. Check the clamp setting.
6. Check the ejector stroke and return.
7. Check to ensure that lowpressure mold protection is set at the proper distance.
8. Ensure that the machine water supply is on and available to the heat exchanger, feed throat, mold and other machine areas.

Quick process start-up

The following procedure works well for any velocitycontrolled injectionmolding machine. The following conditions, although generally applicable, are not necessarily optimum for a specific application.

1. Set pack and hold timers to 0.
2. Set pack and hold pressures to 0.
3. Set injection pressure to the maximum setting.
4. Set transfer position to 6 to 7 mm (0.25 to 0.30″).
5. Set injection time long enough to reach transfer position.
6. Set injection speed to achieve a fill time of 0.5 to 1.5 seconds.
7. Set shot size to achieve a short shot.
8. Increase shot size until part is 95% filled.
9. Note the injection pressure required to maintain 95% fill of the part.
10. Set injection pressure to 21 to 28 bar (300 to 400 psi) above the injection pressure deter mined in the previous step; this insures a velocitycontrolled filling. The molded part should still be 95% filled at this setting.
11. Set the packhold pressure to 50% of the injection pressure setting.
12. Set the packhold time to 2 to 3 seconds to ensure that the remaining 5% of the part is filled and that there is enough additional material to compensate for shrinkage.
13. Finetune the packhold time by running a series of test molding cycles. Weigh each part after each test cycle. Continue to increase the packhold time as required until the part weight does not increase. This ensures that gate freezeoff has occurred and that parts have a repeatable, correct weight. Watch out for overpacking, usually indicated by a series of rings around a gate or evidenced by a "punchedin" gate.

Mold filling, packing and screw rotation

The majority of our TPVs are pseudoplastic shear dependent materials. They require mediumfast to fast mold filling rates via high injection speeds and pressure to increase shear through the system, which reduces their viscosity. This gives maximum flow length to wall thickness ratios, which produces a wellfilled part. We recommend the injection speed be mediumfast to fast. This should generate a shear rate of approximately 10^4 to 10^5 sec 1 through the gates.

Mold packing takes place immediately after the filling stage. This compensates for the loss in volume as the material returns to solid density from the melt. Hold pressure then keeps the molten material under pressure until the gate freezes. Some press systems have controls for both of these phases, and some just have hold phase controls. The user must then configure the single control to accomplish both actions within this constraint.

Cushion requirements

We recommend a small cushion, typically 3 to 6 mm (0.125 to 0.250″) for good cavity packing.

Injection pressures

The actual injection pressure depends on many variables such as melt temperature, mold temperature, part geometry, wall thickness, flow length and other considerations associated with the mold and equipment. Remember that injection pressure is an output from the above mechanical and thermal parameters of the system. The correct injection molding pressure, as discussed earlier, is to obtain sufficient excess injection pressure to allow a velocitycontrolled filling cycle.

Mold packing and holding

Once a mold is 95 to 99% filled, pack and hold time may be optimized. To allow for ample packing, increase the hold time until the gate freezes. Next, adjust hold or pack pressure to ensure full part packing before gate freezeoff. In general, holding pressure should be about 1/2 to 3/4 of the actual injection pressure. See the Quick Process Setup or Scientific Molding Setup Procedure in the Process Startup section for more information.

Screw rotation and back pressure

A high screw RPM (100 to 200) is recommended. Back pressure is not always needed, however, a back pressure of 3.5 to 7.0 bar (50 to 100 psi) may be used to insure a homogeneous melt and maintain a consistent shot size. A higher back pressure is normally employed when using masterbatches.

Estimating molding cycle time

The following guidelines can help to estimate and minimize molding cycle time. These guidelines assume the use of recommended temperatures and processing procedures, and a mold that has a fixed number of cavities plus adequate knock-out area to permit easy part removal.

Molding cycle

Santoprene TPVs from ExxonMobil Chemical allow the use of automatic molding cycles, which eliminates costly manual part removal and reduces the overall molding cycle time by minimizing cooling time. Typically, molded parts form a solid skin and become semirigid, even while their interiors are still molten. This allows parts to be demolded quickly with minimal deformation problems. Obviously, reducing molding time means more costeffective processing.

Oversized runners can be the biggest time wasters in a system. Mold fill usually takes 0.5 to 2.0 seconds, depending on part volume, runnergate style and size, cavity location and injection pressure. Pack (hold time) is usually 2 to 3 times the mold fill times, depending on gate size. If larger gates are used, expect a longer molding cycle time. The speed of an injection molding machine and the part geometry determine the times of other molding cycle phases such as mold close, mold open, ejection and part removal.

Cooling time

Cooling time can be estimated using the equa tion in Table 1.

Table 1
Cooling time (tc)

$tc = (s^2/a) \times T\ factor$
$tc = (s^2/a) \times (1/\pi^2) \times \ln ((8/\pi^2) \times ((Tm\ \ Tw\ ) / (Te\ \ Tw)))$

Symbol	Description	Unit
tc	Cooling time	s
s	section = wall thickness	m
π	Pi = 3.14159	
ln	logarithm base e	
Tm	Temp. melt	°C
Tw	Temp tool wall	°C
Te	Temp ejection	°C
a	Thermal diffusivity = ratio of thermal conductivity to volumetric heat capacity	m^2s^1
$a = \frac{\text{thermal conductivity}}{(\text{density} \times \text{specific heat})} = \frac{W/m\ °C}{((kg/m^3) \times (J/kg\ °C))} = \frac{m^2}{s}$		

Source: How to make injection molds:
Hanser-ISBN: 3-466-16305-0 Chapter 8.1 / pg 272

Cooling time can be estimated by use of the Moldflow Plastics Insight (MPI™) or Moldflow Plastics Advisers (MPA™) programs from Moldflow Pty Ltd. Also, the equation above can be iterated manually.

Cooling complex and thicker parts

If a part has deep undercuts that measure more than 1.6 mm (0.0625″), increase the cooling time to ensure demolding without deforming the part. If a mold has untapered cores as long as 100 mm (4″), decrease the cooling time ; otherwise, the part may shrink tightly on the core, which could cause deformation when the core is pulled.

Drying of the material

Most of our general purpose and injection molding grades are hygroscopic. More details are available from the Product Data Sheet for each grade and/or the Quick Processing Reference. This includes detailed information on processing setup and unique features. Both documents are available on **www.santoprene.com**

Effective moisture control ensures high quality parts that have an appealing surface appearance. It is recommended for these grades that the moisture be reduced to 0.08% or less before processing. Specialized grades may have different drying requirements. These will be spelled out on the Quick Processing Reference for that grade. With any TPV, moisture impacts processing, physical properties, product performance, appearance, or a combination of these properties. Specifically, moisture can cause poor melt quality, splay, voids, porosity, rough surfaces, uneven appearance and/or silver streaking. For some of the specialty bonding grades, drying is essential to prevent material degradation.

Recommended best practices for drying

1. If possible, tumbleblend colorants with the thermoplastic pellets before drying in a desiccant/dehumidified hopper dryer or vacuum oven. If colorants or additives are metered at the hopper, predry colorants separately, unless not recommended by the colorant supplier.
2. Open the drying hopper and make sure it is free of material. Clean the hopper as needed. Be sure to follow standard safety guidelines. Close the hopper and power it up.
3. Set the temperature control to ensure that air into the hopper inlet is at the set point. We recommend using 80° C (180 ° F) for most general purpose and injection molding grades. Some specialty bonding grades have different drying require ments. Please refer to the Product Data Sheet for each grade or Quick Processing Reference for detailed recommendations.
4. Once the dryer temperature is steady, check the dewpoint indicator to ensure that a dew point temperature of 18 °C (0 °F) or lower exists. Fill the dryer hopper. Be sure to record the material, grade, lot number, date and drying startup time. As a suggestion, duplicate the same information on an index or note card and attach it to the side of the dryer.
5. Dry virgin TPV pellets at least three hours, or dry blends of virgin and regrind for four hours.
6. Once the drying cycle is complete, record the finished drying time and move the TPV material to the processing equipment. A best practice would be to have the drying hopper integrated with the injection molding machine. This insures that moisture cannot return to the dried material.
7. To eliminate the chance of an undried material “dead zone” in the dryer, recycle some material from the bottom to the top of the hopper.
8. A best practice would be to measure and confirm the moisture level with a moisture analyzer. For best results, the moisture level should not exceed 0.08%.
9. After the dryer is empty, power it down and again clean the hopper.

Coloring

Parts molded with Santoprene TPV from ExxonMobil Chemical can be produced on conventional molding equipment in any color to match or com plement a finished product. Simply blend neutral Santoprene TPV pellets with the appropriate colorant to achieve nearly any color or hue desired. Precolored TPV pellets are available from several sources.

Solid color concentrates

Solid color concentrates are a versatile, pelletized form of a colorant. These are widely used in the plastics industry due to their stability, compatibility and dispersibility in almost any resin. Color concentrates commonly consist of a pigment compounded with a carrier resin such as polypropylene or polyethylene. Additionally these are well known for allowing good control of color intensity in daytoday consistency. They are also dustfree and easily used. Color concentrates can be either tumbleblended with the base resin or accurately metered into a batch prior to processing. Color changes normally are quick while minimizing cleanup.

The addition of any carrier resin can affect material properties, including hardness, and it may slightly affect the material processing and can affect part shrinkage. Some pigments also affect these properties. Carrierless color concentrates are also available to minimize property and processing effect.

Please follow the individual manufacture’s recommendations for loading. Typically this is about 1 to 5 weight percent, depending on the application and the final color required.

Colorant carriers

We recommend using polyolefinic carriers such as polypropylene with most colorable grades. The specialty bonding grades may require specific or unique carrier or carrier-less systems. Please refer to the Quick Processing Reference for these recommen dations. Never use incompatible carriers. An incompatible carrier can cause problems with melt quality, which is evidenced by delamination in high shear regions or by discoloration. Also, do not use polyvinyl chloride (PVC) as it is not stable at normal TPV processing conditions.

As noted in the drying section above, most colorants absorb moisture, so always follow proper drying procedures unless it not recommended by the colorant supplier.

Carrier-less color concentrates

Carrierless color concentrates are similar to standard color concentrates except there is no plastic resin used to disperse the pigment. A small quantity of binding agent is used. Typically, there is a higher loading of pigment in these systems, and thus less concentrate is used to achieve the same final color as standard concentrates. Also, as no carrier resin is used there is virtually no alteration of physical properties or processing except as caused by the pigment itself.

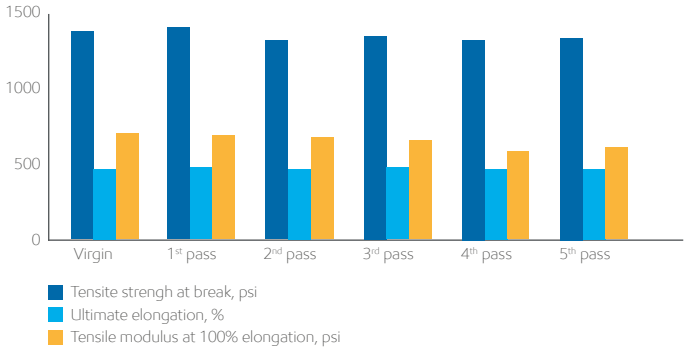
Regrinding

Santoprene TPVs allow for regrinding and reprocessing of clean runners, sprues and scrap parts with minimal variation in material properties. Our materials can be exposed to multiple heat histories with minimal change in tensile strength, modulus, elongation and other properties. Although regrind levels up to 100% can be successfully employed. We recom mend using a consistent percentage of regrind, i.e., 10 to 60%. The ability to use higher regrind levels depends on factors such as machine-part configuration, uniformity of the virginregrind blend and adherence to good housekeeping practices. Higher percentages of regrind (above 40%) tend to cause feeding problems. It is industry best practice to use hopper or feed-throat magnets to capture ferrous metal contami nates from other manufacturing operations.

The following figure (see Figure 4) shows an example of properties retention. This chart is based on Santoprene 20180 TPV processed 6 times through a typical injectionmolding machine. Virgin material was processed in the first pass. Each subsequent pass contained 75% virgin material and 25% regrind from the previous shot. Thus, the last pass contained regrind in proportion from every previous pass. Injection molded plaques of 2 mm were molded and test specimens were taken. Tensile strength at break (psi), ultimate elongation (%) and tensile modulus at 100% elongation (psi) were measured using an ASTM Die C specimen using Test Method TPE0153 (derived from ASTM D 638). Also tested was 15 second delay Shore A Hardness per Test Method TPE0189 (see Table 2) and viscosity, Linear Capillary Rheometer (LCR) at 1200 sec1 per Test Method TPE0200 (see Figure 5).

Always dry regrind before processing. Please refer to the Product Data Sheet for each grade or Quick Processing Reference for these recommendations.

Figure 4
Tensile properties of Santoprene 20180 TPV
(25% regrind material / 75% virgin material)



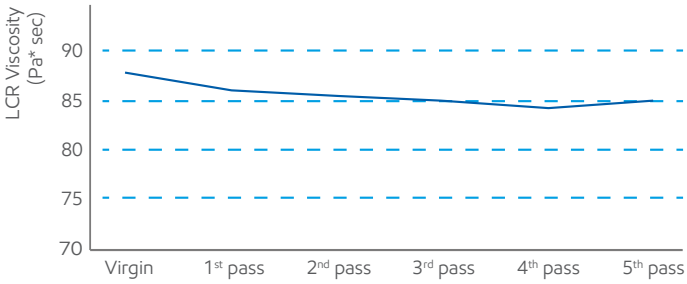
Santoprene 201-80 TPV, lot #PGF1369, per Test Method TPE-0153

Table 2
Hardness change of Santoprene 20180 TPV after multiple molding cycles (25% regrind material / 75% virgin material)

	Virgin	1st pass	2nd pass	3rd pass	4th pass	5th pass
Hardness, Shore A	84.9	84.7	84.9	85.3	85.6	85.1

Santoprene 201-80 TPV, lot #PGF1369, per Test Method TPE-0189

Figure 5
LCR Viscosity at 1200 sec1
(25% regrind material / 75% virgin material)



LCR of Santoprene 201-80 TPV through multiple cycles per Test Method TPE-0200

Shrinkage

For data on shrinkage, please contact the Answer person℠ at **tpe.answerperson@exxonmobil.com**

MOLD DESIGN

Mold steel selection

The specifics of the considered injection molding application determines the type of steel to use in a mold. Table 3 matches certain applications with an appropriate steel grade.

Usually, structural mold sections are made from mediumcarbon SAE 1030 or AISI 4130 steel. Since Santoprene TPVs are generally noncorrosive and nonabrasive, we recommend P20 steel for base plates.

Some TPVs, however, can corrode untreated tool steel after long term contact. In this case, specify H13 steel or use a protective mold coating. Alternately, a corrosion resistant steel like H13 or AISI 420 can be used.

Mold surfaces

The mold cavity finish determines the surface finish of a part. Typically, an SPI B3 to SPI D1 finish (VDI 3036 / Ra 36), provides good mold release for TPVs ; however, if a highpolish, SPI A1 or A2 finish (<3Ra / 30 VDI) is used, an air assisted ejection is required for easy part removal and minimal distortion. When texturing a surface, we suggest a minimum draft of ½° per side, plus ½° per each 0.025 mm (0.001") depth of texture used.

For mold construction, obtain more information about mold steel selection from steel suppliers and moldbase manufacturers. They can provide more detailed information about the steels listed in table 4.

Table 3
Typical high grade mold steels from applications for typical mold steels

Type of steel	Typical uses in injection molds
SAE 1030 or AISA 4130	general mold base plates
P-20	highgrade mold base plates not recommended for cavities, cores, slides and interlocks
420 Stainless steel	best grade mold base plates (no plating required), large cavities, cores and inserts
H-13 (nitrided)	cavities, cores, inserts, ejector pins and sleeves (nitrided)
S-7	cavities, cores, inserts and stripper rings
A-2	small inserts in highwear areas
D-2	cavities, cores, runner and gate inserts for abrasive plastic materials

Table 4
Mold surface plating and treatments

Process	Material	Applied to	Purpose
Coating by impingement molecularly bond	Tungsten disulfide	Any metal	Reduce friction in metaltometal wear with a dry film, which is nonmigrating
Coating by impingement organically bond	Graphite	Any metal	Reduce sticking of plastics to mold surface - migrates
Electroplating	Hard chrome	Steel, nickel and copper alloys	Protect polish and reduce wear and corrosion
Electroplating	Nickel	Steel and copper alloys	Resist corrosion, improve bond under chrome, build up and repair worn or undersized molds
Electroless plating	Nickel	Steel	Protect nonmolding surfaces from oxidation
Electroless plating	Phosphor nickel	Steel and copper	Resist wear and corrosion
Nitriding	Nitrogen gas or ammonia	Certain steel alloys	Improve corrosion resistance, reduce wear and galling; alternative to chrome and nickel plating
Liquid nitriding	Patented bath	All ferrous alloys	I mprove lubricity and minimize galling

Mold plating and coating

The working area of a mold can/should be plated, coated and/or heattreated to resist wear, corrosion and release problems. Table 4: Mold surface plating and treatmens lists treatments that can help reduce wear on gates, runners, ejector pins, core pins, inserts and particularly cavities opposite a gate.

Note: Plating and coating protects only the surfaces of a mold ; heat-treating affects the physical properties of the entire mold.

Mold venting

When molten Santoprene TPV enters a mold quickly under high pressure, enclosed air must exit just as quickly. Sufficient mold venting is essential for efficient processing of TPVs. Even if proper conditions exist throughout a molding cycle, inadequate venting can cause "short shots" parts with burn marks, sink marks, poor surface appearance and low weldline strength. Additionally, without the proper venting, the following can occur:

- Trapped air
- Slow fill time
- Voids
- Shorts Shots
- Poor weld lines

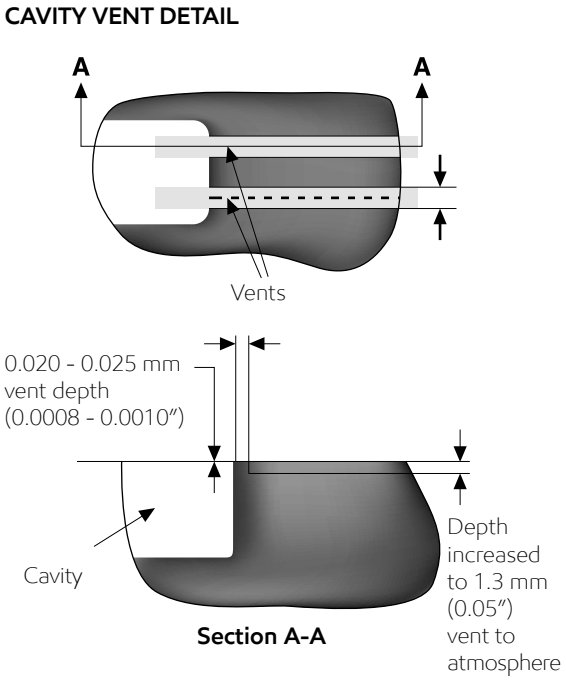
Santoprene TPVs require generous venting as the filling speed is usually very fast (between 0.5 2 seconds). Vents should be positioned at the final fill points or on the periphery of a mold cavity. Long runner segments should be vented separately from the cavity. Soft grades of Santorpene TPV are more susceptible to ventrelated problems. Additional best practices would include:

- All split lines should have peripheral vents.
- All blind grooves or pockets should be inserted.
- Vent and or air blast on ejector pins.

Venting dimensions

Mold venting should provide fast, efficient air removal without interfering with the molding process. Vents usually extend from the cavity to the mold plate's outer edge. Vents should be 0.02 to 0.04 mm (0.0008 to 0.0015") deep.

Figure 6
Typical vent design



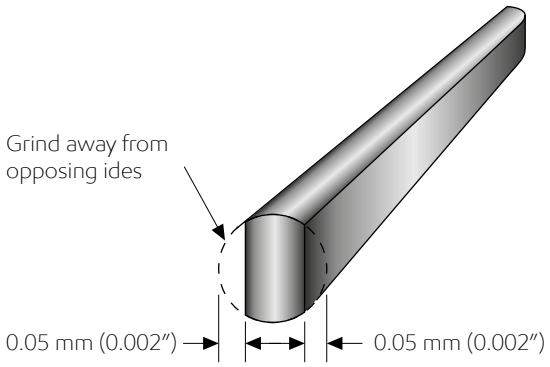
Vent locations

Always place a vent on the mold parting line at a point farthest from a gate to ensure complete mold filling. Additional vents may be needed, depending on the specific configuration of a part. For simpler, generalpurpose molds, locate vents at every spot where knit lines occur. When necessary, use full peripheral vents.

For improved mold filling, vent the runner system as well. For ribs or other restricted areas, drill a hole through each rib top on the mold and plug it with a flatsided pin. This technique is less than ideal since these pins are not selfcleaning and could clog. For deep mold cavities, use strategically located vent pins or ejector sleeves. You can flatten an ejector pin by grinding off opposing sides. This type of vent pin is also selfcleaning during the ejection stroke (see Figure 7).

Figure 7

Flattened vent pin



Sprue, runner and gate design

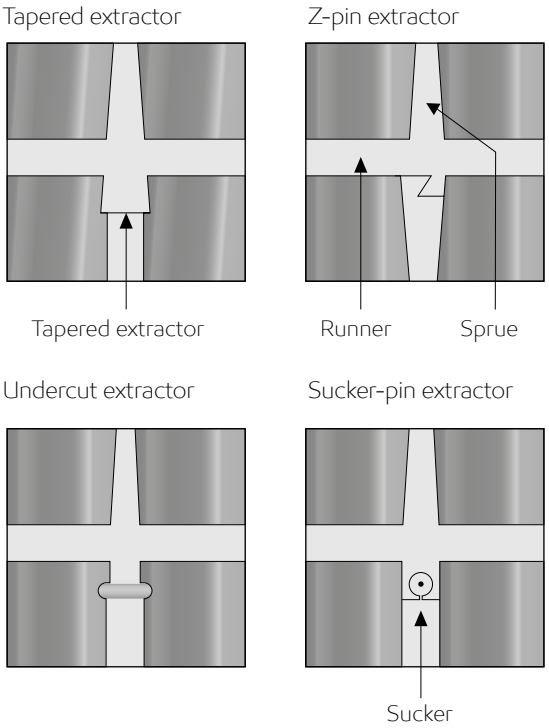
Sprues

They promote a smooth melt transition to the runner and permit easy extraction while providing an optimum pressure drop.

A standard sprue bushing with a minimum taper per side approximately a 2.5 to 3.5 ° included angle (see Figure 8).

Figure 8

Typical sprue extractors



Runner types

Fullround runners have the smallest surfacetovolume ratio and are the most efficient crosssection for pressure transmission and heat retention. This type of runner is cut identically into the 2 mold halves so machining must be precise.

Trapezoidal runners also deliver efficient melt flow. This type of runner system is machined into a single mold plate. Runner walls should have a 5° taper with a round runner bottom. A modified trapezoidal runners mold is shown in the Runner Types figure (see Figure 9).

Halfround runners are not recommended since they give the lowest flow and are the most easily overcooled of all runner types in common use.

Figure 9
Runner patterns

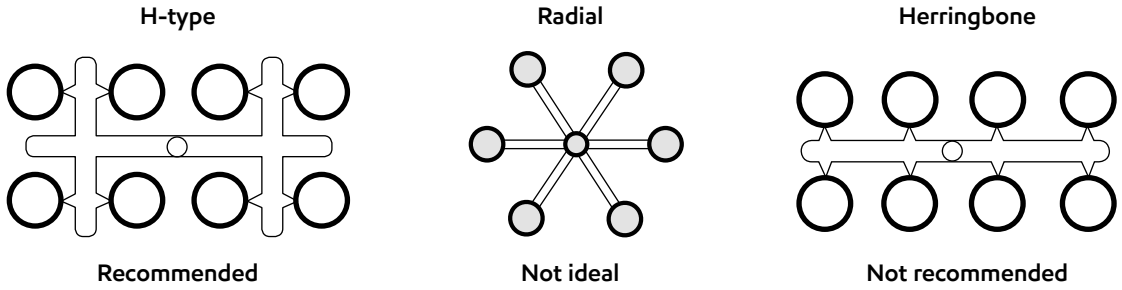
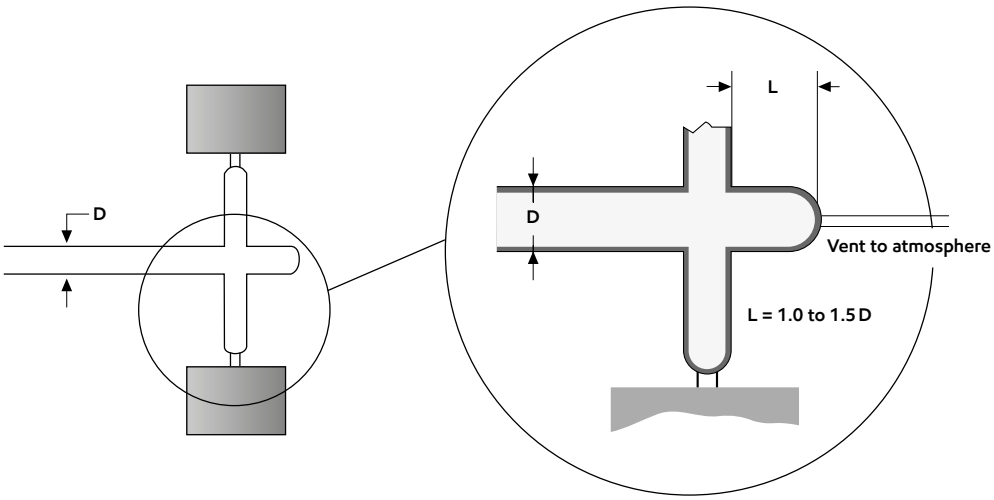


Figure 10
Cold slug well size and location



Cold slug wells

Cold slug wells (see Figure 10) capture the cooled material found in the nozzle tip and the material that is cooled during the runner filling. If injected into a part, a cold slug can lead to surface imperfections. A cold slug well captures solidified material and prevents it from interfering inside that cavity. Additional wells should be used at the end of each runner section (turn/split).

Gating

Gates play a number of crucial roles in injection molding.

Santoprene TPVs from ExxonMobil Chemical have a relatively high melt viscosity at low shear rates. Viscosity decreases as the shear rate increases.

Increasing temperature has little effect on TPV melt viscosity. Smaller gates and higher shear rates keep melt viscosity low and improve melt flow.

As a general rule, minimize the number of gates per cavity. One gate per cavity is usually best to avoid weld lines and crossflow patterns ; however, some applications may require multiple gating. For more information contact the Answerperson℠ at: tpe.answerperson@exxonmobil.com or visit us at **www.santoprene.com**

Gate location

Gate location(s) affects the mold filling pattern and flow orientation ; gate location is just as important as gate size and type. Locate gates to ensure rapid and uniform mold filling. As a general guideline and recommendation:

1. Gates should be located to feed the thickest section of the part first. Material will flow from thick to thin sections. This promotes uniform flow and allows good packing.
2. Optimum gate locations will direct the melt flow through the length of the part with nearly equal flow distances to all its edges.
3. Place the gate in an unstressed (nonfunctional) area of the part. Gate areas often contain high residual stresses from the filling process.
4. Gates must be located to direct trapped air towards the vents. Locate venting opposite the gate at the farthest flow point (see Mold venting section for more information).
5. Avoid gating locations that will cause melt fronts to converge or backfill (race track), thus entrapping air.
6. Locate a gate so that it directs material towards the wall or a pin instead of into a free area. This will cause the material to disperse and continue to flow instead of jetting.
7. Position the gate to minimize weld line problems, or to move weld lines to preferred areas.
8. To reduce the number of multiple weld lines and venting problems, use as few gates as possible to fill the part.
9. Use centergating for round or cylindrical parts to maintain concentricity.
10. Gate land lengths should be kept short as possible.
11. Gate size should be kept small. A gate shear rate in the range of 10^4 to 10^5 sec^1 is recommended.

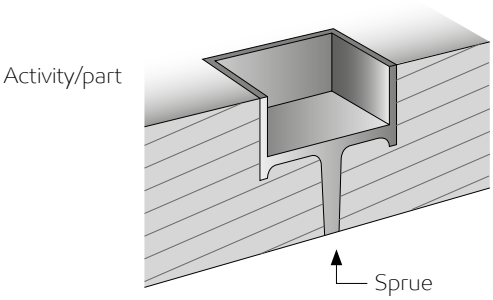
Gate design

Virtually any of the gating styles typically used in the injection molding industry can be used to successfully process Santoprene TPVs. As with the gate style, it is important to select a gate type that is appropriate for the specific part geometry.

Direct sprue gates

Direct or sprue gates (see Figure 11) may be used with single cavity mold where the sprue feeds material directly into the cavity. If possible, provide a cold slug well opposite the sprue gate.

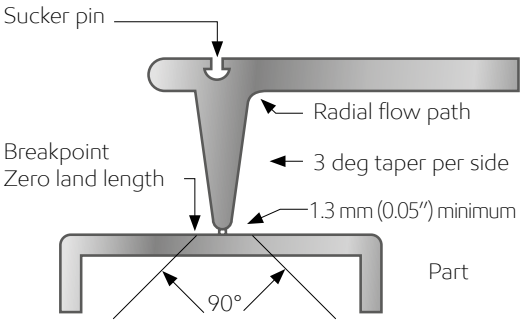
Figure 11
Direct sprue gate



Pin gates

Pin gates (see Figure 12) are actually small gates located on the parting line to minimize gate vestige/blemish. They are especially popular for automatic degating in threeplate molds. Another advantage of pin gates is that they can easily provide multiple gating to a cavity (for thinwalled parts).

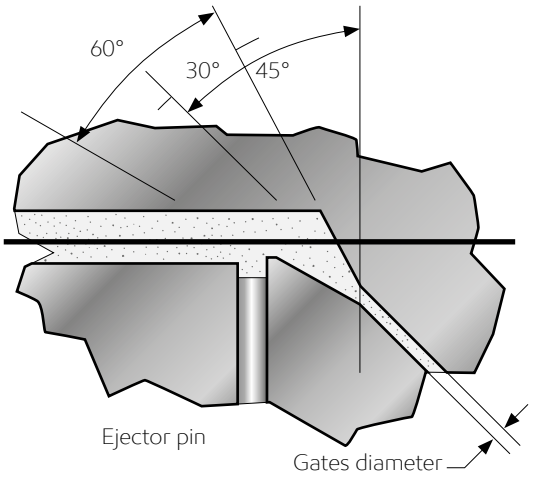
Figure 12
Pin gate



Submarine gates

Submarine gates provide automatic degating of a part from the runner system during part ejection. Submarine gates enter the mold below the parting line. To reduce pressure loss and prevent premature gate freezeoff, submarine gates should be as short as possible. The typical size range is 0.80 to 1.5 mm (0.03 to 0.06″). A sharp angle should range from 20 to 30 °, with a 60 ° angle from the runner. Figure 13 illustrates a suggested gate configuration. Unless special drilling is used, the gate will be oval due to the angle of entry.

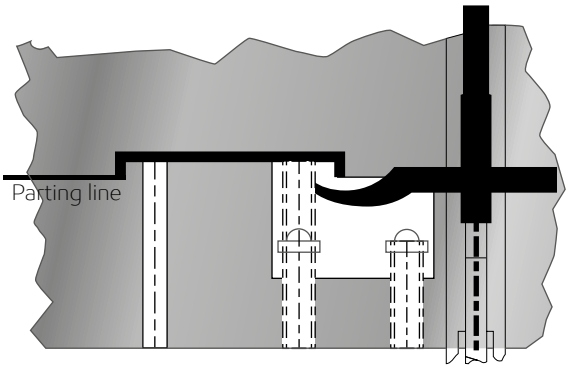
Figure 13
Submarine (tunnel) gate



Curved submarine gates

Cashew gates move the gate vestige from the side of a part (often an appearance surface) to somewhere underneath the part where it can be more tolerated. The following figure shows a schematic of this type of gate.

Figure 14
Curved submarine (cashew) gate



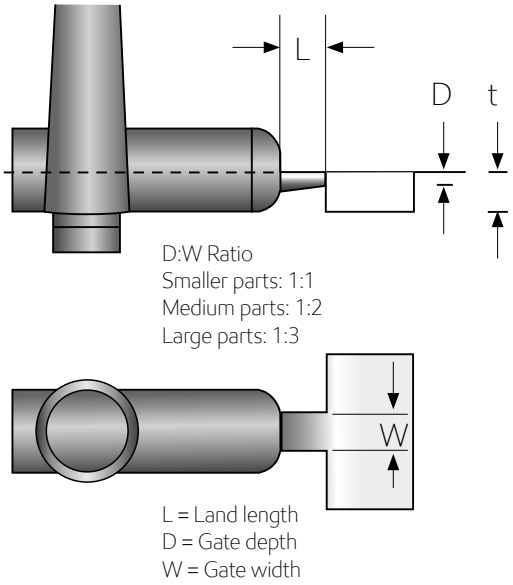
Ring and diaphragm (disc) gates

A diaphragm or disc gate is used for cylindrical parts that require good concentricity and elimination of weld lines (for high strength). The gate land or membrane should be significantly thinner in crosssection than the runner ring or central disk. The thickness differential forces the ring or disk to fill completely before the melt fills the membrane. This type of gate requires a post mold degating operation.

Edge gates

Edge gates, both rectangular and round, must be large enough to avoid melt overheating due to frictional heating. The preceding figure details typical depthtowidth ratios for edge gates.

Figure 15
Edge (rectangular) gate

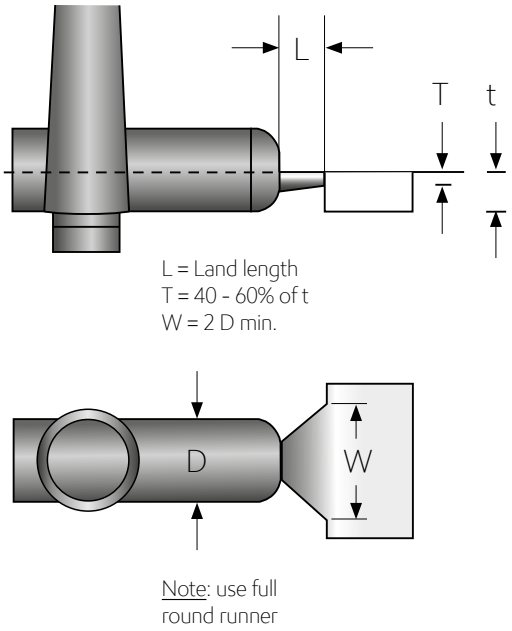


Fan gates

For flat mold sections, a modified fan gate may be used, which minimizes jetting and significantly reduces the high stresses that occur during mold packing. The plastics industry suggests smooth radii and transitions between runners and gates. Fan gates can be balanced so they distribute the melt evenly across the land before the melt enters the mold cavity. Fan gates require a post molding degating operation and may leave noticeable vestige.

Figure 16

Fan gate



L = Land length
T = 40 - 60% of t
W = 2 D min.

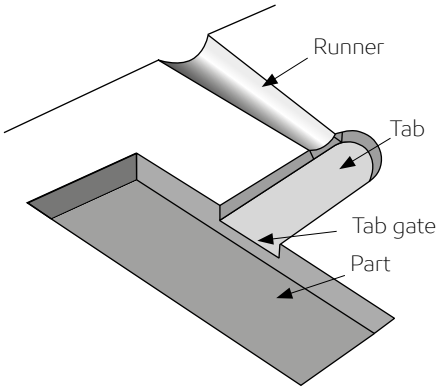
Note: use full round runner

Tab gates

Tab gates provide a uniform melt orientation when an application requires a large volume for mold filling. Tab gates help reduce the effect of residual stresses, gate blush and jetting in the gate area. They are used where flatness is critical or in large surface areas that may have a tendency to warp. Tab gates require a post molding degating operation and may leave noticeable vestige.

Figure 17

Tab gate

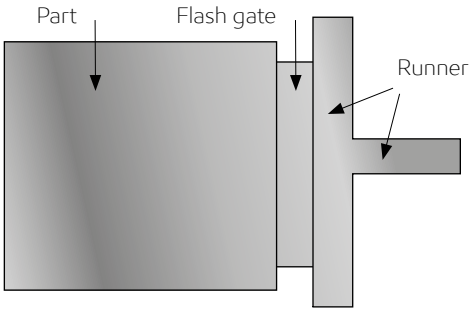


Flash gates

A flash gate is used to minimize warping in flat or very large parts. Flash gates extend across a part from 0.38 to 0.76 mm (0.015 to 0.030") deep and have a chisel taper across the gate land length. Flash gates require a post molding degating operation and may leave noticeable vestige.

Figure 18

Flash gate



Ejection

The ejector system design and method should be based on a particular product and its hardness. Ejector system design method and selection recommendations follow:

1. To demold soft grades of Santoprene TPV, a combination of stripper plates and integrated air blast is required.This is essential when it comes to parts with deep draw and heavily undercuts.
2. To demold hard grades of Santoprene TPV, ejector pins are required. The pins should always be as large as possible and push against supported sections to avoid part deformation.
3. To make part demolding easier, sand or vapor blast the mold cavity walls with a matte or satin surface finish SPI #3 or VDI 3336 reference 3400.
4. Use draft to ensure troublefree part ejection. On cavity and core sides, an angle of 0.25 to 1° per side is normally sufficient for the majority of injection molded parts. Bosses and ribs should be molded with a minimum of 3° draft. Refer to the Quick Processing Reference for any special draft considerations.

The inherent elastic properties of Santoprene TPVs make it possible to demold parts of diverse design and complexity without damaging them when they are stretched or flexed. Parts with undercuts that would normally require side cores, slides, collapsible cores and other mold design accommodations associated with hard thermoplastic materials can be demolded easily using a more simplyconstructed mold.

Undercut design guidelines

Proper part and tool designs are even more critical if a molded part has undercuts. The following recom mendations identify key undercut design conside ra tions:

1. Fillet and taper an undercut well to allow for expansion and slippage ; this avoids tearing at sharp corners during part ejection.
2. The undercut part must be free to stretch or compress ; that is, the part wall opposite an undercut must clear the mold or core before part ejection starts. Use a matte, beadblasted steel finish (SPI #3 VDI 27 or coarser) for cavity and core inserts. This finish eases part release during ejection.
3. Air breaks the vacuum that exists between a part and the undercut core before the mechanical ejector takes over.

Mold temperature control

The cooling time for a molded part is typically 60 to 80% of the total cycle time. Uniform and efficient mold temperature control is critical for minimizing cooling time, maximizing production rate and controlling part tolerances. Adequate cooling is one of the most important aspects of mold design. The cooling system should have the ability to rapidly and uniformly cool a mold. It also should be able to maintain mold and system temperature settings in a production environment from cycle to cycle. Uniform cooling not only ensures a shorter molding cycle, but also minimizes differential shrinkage and reduces internal stresses.

HOT RUNNER SYSTEMS

Hot runner tooling techniques can be used to efficiently process Santoprene TPVs in a variety of commercial applications. In fact, in most cases, hot runner processing can actually improve part quality and reduce molding cycle time.

Hot runner systems have runners that are internal to the mold, and the system maintains material in the runner at about the same temperature as the barrel. The benefits of hot runner systems are that there are no runners or gates to be trimmed and recycled, and cooling the runner does not require additional cycle time. In addition, smaller gates can be used to improve molded part appearance.

System selection

For successful processing, a hot runner system should keep the material melt temperature at or near nozzle temperature as the material passes through the gate and into the mold cavity. These four steps are recommended:

1. Minimize the time material remains in a hot runner system. If possible, it should contain no more than one shot.
2. Design melt channels for uniform pressure drop throughout the system.
3. Prevent or eliminate possible "hangups" (i.e. burrs, sharp corners and heating rods) and other melt flow interruptions.
4. Maintain precise temperature control and make sure heat is evenly distributed throughout the system, including drops and manifolds.

Externally heated manifold

An externally heated manifold contains a manifold block with a hollow interior that channels the main runner flow. Heaters parallel the runner flow channel and heat the entire manifold block. Drops or auxiliary channels are usually heated as well and feed the melt into the mold cavity.

We recommend using this type of manifold system for hot runner tooling since it provides a "straightthrough" melt flow path. It minimizes pressure drop and reduces the chance of material "hangup," discoloration and burning. In addition, the entire manifold block is heated, making it easier to maintain uniform temperature. An externally heated manifold system requires a sufficient number of heaters and temperature sensors for precise temperature control.

Insulated runner molds

We do not recommend insulated runner molds. Insulated runner molds incorporate an oversized runner and create a skin of cooled material that insulates the melt. As a result, degraded or unmelted material can enter the melt flow. During long cycle interruptions, the entire unheated runner can freeze off and require disassembling the mold for cleaning.

Mold design

Manifolds

In general, hot runner manifolds for most TPVs have reduced flow channel diameters compared to those used for rigid thermoplastics. For most TPV applications requiring the use of hot runners, we recommend smallbore, fullround channels with a 6.4 mm (0.25") diameter. This channel size provides a uniform pressure drop that minimizes material residence time within the manifold. Make sure all material passages are smooth and streamlined without protrusions, sharp corners or blind spots. To take advantage of the melt flow properties of Santoprene TPVs, flow channels should be cylindrical rather than annular.

A balanced manifold system is essential for efficient, multicavity hot runner processing. In each section of a mold, match the length and diameter of the primary and secondary flow channels. Since flow channels in a manifold function as full-round runners, the melt flow must be balanced to uniformly fill all cavities.

Shot size verses manifold volume

Shot size verses the manifold volume determines the residence time. If the diameter of a manifold is too large, the material tends to hang and form channels, which results in an insufficient shear rate and an excessive residence time possibly leading to degradation. Ideally, the manifold volume (runners plus probes) should be significantly less than the shot volume ; however, high residence times with small parts cannot be avoided.

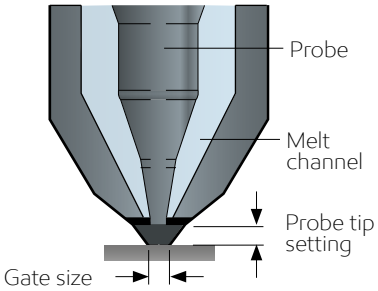
Probes

Gates are an integral part of a heated probe. The probe is attached to the downstream end of the melt channel. The probe fits down the melt channel center, forming an annular flow path for the material. Manufacturers of probes offer various models according to the type of gate: hot tip, edge, valve or core. Remember all probes need to be insulated using DuPont™ Vespel ® or equivalent. An end cap or bushing will restrict the amount of dead material coming into contact with the relatively cold mold steel. This forms a cold slug.

It is essential in any gating system to maintain the melt temperature with minimum temperature loss between the flow channel and the mold cavity. Excessive temperature loss could cause a cold slug to form at a gate. The cold could cause a defect on the surface of a molded part. Probe recommen-dations are summarized below:

1. The probe channel must be provided with an external heater. This can be achieved most efficiently by embedding cartridge heaters in the probe metal. Other methods include heating with a socket (continuous heating) or a coil (discontinuous heating). The choice depends on the type of probe selected.
2. The probe channel should be small enough to achieve a high shear rate.
3. The tip of the probe is often the critical point and any heat loss at this juncture must be avoided. The cartridge heater should be entirely embedded in the metal of the probe. Copper alloys enable an adequate temperature to be maintained through to the gate. In most critical cases, adding a thermoplastic insulator, i.e., DuPont™ Vespel ®, around the tip of the probe further assists flow at this point and should improve surface finish in the gate area.
4. Effective gate diameter at the end of the probe should be approximately 1.0 mm (0.040") for parts with a maximum weight of 50 grams (1.76 oz) ; for parts that weigh 10 grams (0.35 oz) or less, a diameter of 0.8 mm (0.032") is acceptable.
5. A minimum probe clearance between the gate and cavity will help optimize appearance at the injection point (see Figure 19). Thus, a clearance (at ambient temperature) of about 0.2 mm (0.0080") results in a barely visible injection point (gate vestige). For additional details, consult the manufacturer's specifications for hot runners and probes.

Figure 19
Probe clearance



Gate selection

For individual hot runner applications, carefully evaluate part design and gate styles to determine the following:

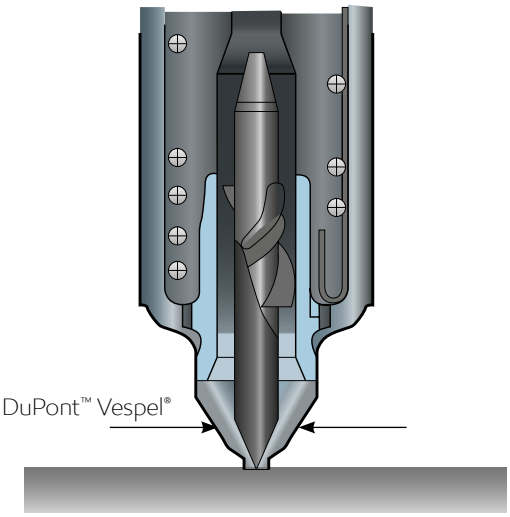
1. Optimum location for gating
2. Number of gates needed to fill a part as quickly as possible (typically 0.5 to 2.0 seconds)
3. Ensure the air and gas is directed to an area of the part where it can be effectively collected and vented during the extremely short filling phase of the process

Hot runner processing requires relatively smaller gates than those used in cold runner processing. In general, hot runner gates range in size from 0.8 to 1.5 mm (0.032 to 0.060"). A gating system should maintain melt temperature with a minimum of heat loss between the flow channel and the cooler mold cavity.

Open / torpedo-tip gates

A torpedotip, or open, gate has a heated torpedo tip or probe which provides a highpressure seal from the hot nozzle to the cooled gate insert and minimizes heat loss. Due to the flow characteristics of most TPVs, clearances can be a problem and occasionally the torpedotip support fins can also create flow problems. The gate diameter should range from 0.8 to 1.5 mm (0.032 to 0.060"), and the probe or the torpedo should be 0.02 mm (0.001") behind the gate and the heat source. If needed, the torpedo can be moved forward into the gate to eliminate vestige. As this gate is always open, there is always a tenancy to drool. This probe design is only recommended for large nonaesthetic parts where gate vestige is not important. For a fully controlled runner system and good gate aspect, valve gate systems are recommended.

Figure 20
Open / torpedotip gates

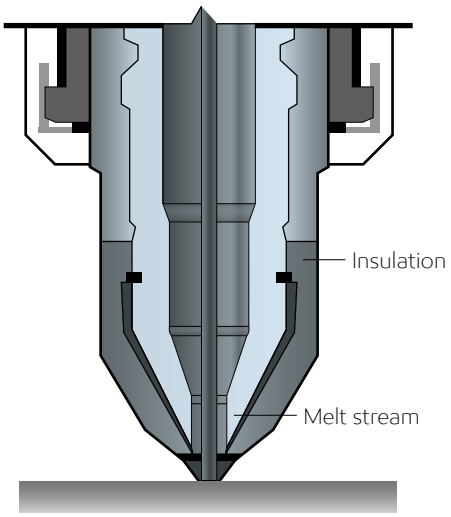


Valve gates

Valve gates are the only gate types recommended for use with Santoprene TPVs. They are the only way to effectively control nozzle / gate drool as well as give a good gate aspect. Valve gates feature a small doubleacting piston in their center that usually is powered by hydraulic or pneumatic pressure. This piston opens during filling and packing, then closes to minimize gating vestige. To avoid damaging the gate area, the piston stem should have a positive stop for closing. Good shutoff virtually eliminates gate vestige on a molded part, thus improving appearance and preventing nozzle drool when the mold opens. They are suitable for both small and large molded components.

For additional details, consult the manufacturer's design requirements for valve gate probes and their installation.

Figure 21
Valve gates



Overmolding

For an even wider variety of part applications, polymers can be combined through several different types of multishot injection molding processes. These include insert molding, twoshot molding/2K or 2component molding, and coinjection molding. We offer TPVs that can bond directly to a variety of substrates.

Standard grades of Santoprene TPV bond to polypropylene, some polyethylenes and some TPOs. Nylonbondable grades of the PA Series of Santoprene TPV bond to nylon 6, nylon 6 with a variety of fillers, and to some blends of nylon 6/6. Only twoshot molding is recommended. Other special bondable grades bond to a variety of engineered thermoplastic (ETP) substrates such as PC, ABS, ASA, PC/ABS, PET, PBT, PS and HIPS. Please use the Advanced Product Data Search tool on the website.

Insert molding

Insert molding (sometimes called overmolding) consists of placing a solid insert (or preform) into the mold and injecting around it. If the insert and the overmolded material are compatible, a melt bond occurs between the two materials at their interface. The strength of this bond is affected by several factors, including interface temperature, cleanliness of the insert and the melt temperature. Bonding can be improved by preheating the insert and ensuring that the surface of the insert is free from dirt, grease, hand oil from an operator or any other contaminant. Noncompatible insert materials such as metals also can be used. This requires that an adhesive be applied to the insert so that it bonds with the overmolded material. Otherwise, a design utilizing mechanical locks is required.

Insert molding is the simplest of the multishot processes since it is most similar to standard injection molding. For low volume applications, the inserts can be handloaded, but for higher volumes, robotic pick and place methods may be a costeffective choice.

Two-shot molding

Twoshot (also known as 2K or 2component) molding consists of a machine having two independent injection units shooting two different materials into the same mold through separate runners and gates. The first material is shot through a primary runner system as in normal molding, while the runner system for the second material is shut off. After the first shot, the mold slides or rotates to shut off the first runner system and opens the second for the next shot of material. After the second shot, the part is cooled and ejected normally.

Twoshot molding requires that the two materials be compatible or no bonding occurs. The bond strength of a twoshot molded part is generally better than that of an insertmolded part due to the higher temperature at the melt interface.

Co-injection molding

Coinjection or sandwich molding differs from twoshot molding because coinjection molding uses a common runner and gate for both materials. This allows for a soft touch material to be on the outside while having a hard, compatible core material to provide mechanical properties to the system. This construction allows the designer to take advantage of the individual properties of each material. For example, consider a heavy-walled part. Cycle time can be substantially reduced by processing the core material at a lower melt temperature (to decrease cooling time) while running the skin material at a higher melt temperature to achieve a smooth surface.

Efficient coinjection requires a separate injection unit for each material. Also, materials can be coinjected into specially designed rotary or shuttle molds.

QUICK REFERENCE NOTES

Screw design
Special equipment is not required use a screw L/D ratio of approximately 20:1 and a compression ratio of 2.5:1.

Barrel capacity
Barrel capacity should be no less than 1.3 shots and no more than 4 shots.

Clamp pressure
Clamp pressure should be 29 to 49 MPa (3 to 5 T/in2) of projected area.

Screw RPM
Screw RPM should be 100 to 200.

Screw cushion
Screw cushion should be 3 to 6 mm (0.120 to 0.250") Refer to the processing section on page 7 of this manual.

TROUBLESHOOTING

Injection molding requires controlling and finetuning many machine and process variables. Problems can arise that result in molded parts that are unsatisfactory. Thus, troubleshooting involves control of the molding process by maintaining the correct adjustment and balance of these major variables.

Varying machine conditions in a logical manner during the molding cycle is an effective approach for trouble shooting and correcting most processing problems. Troubleshooting recommendations in this section frequently contain more than one recommended corrective action, so change only one condition at a time and be sure to allow sufficient time for the system to reach equilibrium at the new condition. If the trouble is not corrected, proceed to the next recommendation. Keep a careful record of each change, noting how it affects subsequently molded parts.

Troubleshooting guide

Problem	Probable cause	Corrective action
BURN SPOTS ON PART	1. Mold is insufficiently vented.	A. Make sure vents are clear of obstructions. B. Insure vents are deep. C. Add vents, if necessary, to assure good operation.
	2. Injection rate is too high.	A. Reduce injection speed.
	3. Screw RPM is too high.	A. Decrease screw RPM.
	4. Back pressure is too high.	A. Reduce back pressure. Minimum is 0.3 MPa (50 psi).
	5. Clamp pressure is too high.	A. Reduce clamp pressure.
BURN MARKS AT GATE	1. Melt flow across restricted gate area causes excess shear heat.	A. Reduce injection speed. B. Reduce back pressure. C. Decrease nozzle temperature. D. Decrease front barrel zone temperature. E. Decrease mold temperature in gate area. F. Check for resin contamination.
DISCOLORED STREAKS OR CHUNKS	1. Additives or colorants have incompatible carriers.	A. Change to an additive or colorant material which has a polyolefin base, such as polyethylene or polypropylene. B. Never use additive or colorant materials which have polyvinyl chloride carriers.
	2. Prior runs have contaminated the plastication system.	A. Physically clean the entire system - hopper, barrel, screw, shut off valve, nozzle and mold. B. Purge the system with polyethylene or polypropylene.
	3. Melt has insufficient color concentrate blending.	A. Increase back pressure. B. Increase barrel temperature. C.Reduce rear zone temperature. D.Change screw RPM. E. Decrease nozzle orifice diameter. F. Replace screw or nozzle with one which has a mixing design.
DISTORTED PARTS	1. Material is oriented during injection.	A. Decrease injection velocity for slower filling. B. Increase mold temperature. C. Adjust velocity profile to provide homogeneous melt.
	2. There is a difference in packing density.	A. Increase hold pressure. B. Increase injection fill speed to fill faster. C. Increase screw RPM.
	3. Molded part has stresses.	A. Increase melt temperature. B. Prevent overpacking at gate by reducing hold pressure or reducing hold time. C. Lengthen cooling time. D. Delay knockout actuation. E. Increase injection forward time. F. Reduce temperature differential in mold.
	4. Material flow into mold is not uniform.	A. With multi-cavity molds, make sure flow is balanced.
	5. Part is too hot when ejected.	A. Reduce mold temperature. B. Increase cooling time. C. Reduce melt temperature.
	6 Knockout is operating improperly.	A. Knockout engaging too fast, slow down ejection. B. Non-uniform operation of pins. C. Bearing area of knockout pins is too small. Increase diameter and number of knockout pins. D. Relocate knockout pins to strong area of part. E. With delicate parts, change to air blow-off.
	7. Part design is incorrect.	A. Make sure part contains no sharp variations in cross sections. B. Correct taper or draft insufficiencies.

Problem	Probable cause	Corrective action	
FLASH ON PARTING LINE OF PART	1. Excessive injection pressure causes mold to part.	A.	Verify injection fill time.
		B.	Decrease final injection speed.
		C.	Reduce back pressure.
		D.	Decrease hold pressure.
		E.	Increase clamp pressure.
		F.	Move transfer point back.
	2. Melt temperature is too high.	A.	Reduce nozzle temperature.
		B.	Reduce front barrel zone temperature.
		C.	Reduce back pressure.
		D.	Decrease mold temperature.
		E.	Reduce screw RPM.
	3. Mold function is incorrect.	A.	Make sure mold faces are in correct alignment.
		B.	Remove foreign matter from mold faces.
		C.	Decrease size of vents.
		D.	Decrease mold temperature.
		E.	Check platens for square.
	4. Mold release agent causing flash on part.	A.	Mold surface lubricant should not be used.
JETTING	1. Mold design has insufficient impingement	A.	Relocation of gate to create impingement.
MATERIAL DROOLS FROM NOZZLE	1. Pressure build up.	A.	Add decompression/suck-back.
	2. Barrel front zone is too hot.	A.	Reduce front barrel zone temperature.
	3. Nozzle is too hot.	A.	Reduce nozzle temperature.
PART DIMENSIONS ARE TOO LARGE	1. Mold is over packed.	A.	Verify injection fill time.
		B.	Decrease hold pressure.
		C.	Decrease overall injection speed.
		D.	Decrease final injection speed.
		E.	Decrease back pressure.
		F.	Decrease front barrel zone temperature.
		G.	Increase mold temperature.
	2. Mold designed incorrectly.	A.	Verify mold shrinkage factor used; adjust mold design if necessary.
	3. Material is wet.	A.	Pre-dry the material.
PART DIMENSIONS ARE TOO SMALL	1. Mold is underpacked.	A.	Verify injection fill time.
		B.	Increase overall injection speed.
		C.	Increase hold pressure and hold time.
		D.	Reduce cushion in cylinder to 6.0 to 3.2 mm (0.250 to 0.125″).
		E.	Increase back pressure.
		F.	Decrease mold temperature.
		G.	Increase melt temperature.
		H.	Increase cavity venting.
		I.	Increase cooling time.
	2. Mold design is incorrect.	A.	Verify mold shrinkage factor used; adjust mold design if necessary.
		B.	Add additional gates.
PARTS STICK IN MOLD CAVITY OR SPRUE	1. Part is not sufficiently cooled.	A.	Reduce stock temperature.
		B.	Decrease second stage injection time.
		C.	Reduce injection pressure.
		D.	Reduce screw RPM.
		E.	Add decompression.
		F.	Increase overall cycle time.
		G.	Decrease mold Temperature.
	2. Mold design is incorrect.	A.	Seal nozzle correctly in sprue bushing.
		B.	Reduce length of sprue.
		C.	Increase draft on side walls of part to assure smooth release.
		D.	Draw polish cavity surface in direction of ejection. (Consult the Answerperson℠ before implementing this corrective action.)

Problem	Probable cause	Corrective action
PARTS STICK IN MOLD CAVITY OR SPRUE (cont.)	3. Cores are too slender.	A. Make cores stronger to prevent misalignment by injection pressure.
	4. There is an overpack problem.	A. Reduce injection speed. B. Back off transfer point. C. Reduce pack/hold pressure.
POOR FINISH	1. Mold fill is too slow.	A. Increase injection pressure. B. Increase injection speed. C. Increase melt temperature. D. Increase gate size.
	2. Mold function is incorrect.	A. Make sure venting is adequate. B. Increase mold temperature. C. Do not use mold lubricants. D. Clean cavity surfaces.
	3. Part design is incorrect.	A. Reduce abrupt changes in section thicknesses. B. Avoid ribbing that is too thick.
POOR SURFACE DETAIL	1. Mold is filling improperly.	A. Increase shot size. B. Increase overall injection speed. C. Increase back pressure. D. Increase mold temperature.
	2. Packing conditions are improper.	A. Increase injection fill time. B. Increase mold temperature. C. Increase second stage pressure and Time.
POOR WELDS ON PART	1. Material flow is cooling in cavity too quickly.	A. Increase melt and mold temperature. B. Increase injection pressure. C. Increase ram speed. D. Increase nozzle temperature. E. Lengthen cycle time.
	2. Mold design and/or function are improper.	A. Make sure vents are not plugged. B. Increase gate size; relocate for most efficient melt flow. C. Raise mold temperature. D. Increase size of runner. E. Decrease fill speed.
SHORT SHOTS PREVENT COMPLETE MOLD FILLING	1. Available melt volume is insufficient.	A. Increase shot size; confirm a cushion of 3.2 mm (1/8”). B. Increase injection fill time. C. Make sure nozzle and runners are clear of obstructions. D. Increase front barrel zone temperature. E. Increase nozzle temperature. F. Verify operation of non-return check valve on screw.
	2. Mold has high pressure drop.	A. Increase overall injection speed. B. Increase back pressure. C. Increase hold pressure. D. Try a higher flow grade. E. Add gates.
	3. Mold function is incorrect.	A. Increase mold temperature. B. In multi-cavity molds, be sure all gate sizes are uniform to prevent unbalanced melt flow. C. Check part design to make sure there are no restrictions. D. Make sure mold is vented correctly and vents are clear.
SILVER STREAKING	1. Material has trapped moisture.	A. Pre-dry material according to instructions. B. Make sure mold is vented properly.
	2. Melt temperature is too high.	A. Reduce barrel heating to prevent decomposition. B. Reduce screw RPM to stop excessive shear heating.
	3. Cold mold causes moisture condensing.	A. Increase mold temperature; make sure cavity is dry.
SINK MARKS	1. Mold is underpacked.	A. Increase overall injection speed. B. Increase boost cut off set point. C. Increase hold pressure and temperature. D. Increase cushion in cylinder.

Problem	Probable cause	Corrective action
SINK MARKS (CONT.)	1. Mold is underpacked. (Continued)	E. Increase shot size. F. Increase barrel heating. G. Increase back pressure. H. Add or increase nozzle heating.
	2. Mold function is incorrect.	A. Be sure gates are large enough and are correctly located. B. Reduce length of sprue bushing, increase bushing diameter. C. Increase mold temperature. D. Make sure vents are large enough and are clear. E. Increase mold cooling cycle time.
	3. Part design is incorrect.	A. Redesign part to remove sharp thickness changes in section.
SPLAY MARKS IN GATE AREA OF PART	1. Last part of mold fill has slow pressurization.	A. Increase mold fill speed. B. Increase back pressure. C. Increase second stage injection time. D. Increase barrel temperature. E. Increase mold temperature. F. Increase screw RPM. G. Remove decompression.
	2. Mold set up is not correct.	A. Make sure sprue and runner system are direct and sized correctly to ensure fast, efficient flow. B. Shorten land length of gate.
VOIDS IN MOLDED PART	1. Material in core sets up more slowly than surface material, especially in thick cross sections.	A. Increase injection pressure and time. B. Increase back pressure. C. Increase mold temperature. D. Make sure sprue, runner and gates are large enough to assure free flow. E. Increase cooling time.
	2. There is moisture in pellets.	A. Pre-dry pellets according to instructions.
WARPING	1. See Distorted Parts above.	A. See DISTORTED PARTS problem above for corrective action.

ExxonMobil

Energy lives here™

Contact us for more information:
exxonmobilchemical.com

S0817-030E49

©2017 ExxonMobil. ExxonMobil, the ExxonMobil logo, the interlocking "X" device and other product or service names used herein are trademarks of ExxonMobil, unless indicated otherwise. This document may not be distributed, displayed, copied or altered without ExxonMobil's prior written authorization. To the extent ExxonMobil authorizes distributing, displaying and/or copying of this document, the user may do so only if the document is unaltered and complete, including all of its headers, footers, disclaimers and other information. You may not copy this document to or reproduce it in whole or in part on a website. ExxonMobil does not guarantee the typical (or other) values. Any data included herein is based upon analysis of representative samples and not the actual product shipped. The information in this document relates only to the named product or materials when not in combination with any other product or materials. We based the information on data believed to be reliable on the date compiled, but we do not represent, warrant, or otherwise guarantee, expressly or impliedly, the merchantability, fitness for a particular purpose, freedom from patent infringement, suitability, accuracy, reliability, or completeness of this information or the products, materials or processes described. The user is solely responsible for all determinations regarding any use of material or product and any process in its territories of interest. We expressly disclaim liability for any loss, damage or injury directly or indirectly suffered or incurred as a result of or related to anyone using or relying on any of the information in this document. This document is not an endorsement of any non-ExxonMobil product or process, and we expressly disclaim any contrary implication. The terms "we," "our," "ExxonMobil Chemical" and "ExxonMobil" are each used for convenience, and may include any one or more of ExxonMobil Chemical Company, Exxon Mobil Corporation, or any affiliate either directly or indirectly stewarded.