Santoprene™ TPV

ExxonMobil

TPV troubleshooting guide for injection molding

Energy lives here™

CONTENT

The following table provides an overview of most problems/defects. Please click on the desired problem phrase for a quick view of problems and potential causes:

TROUBLESHOOTING

PROBLEM: BURN MARKS AT GATE

Generally burn marks at the gate are caused by an over-shearing of material. The melt flow will reach its highest shear rate while in the gate. Since the gate will also be the hottest area, over-shearing here will leave a burn mark at the gate.

Gate size
Using smaller gates and higher shear rates keeps melt viscosity low, which improves melt flow of our TPV materials. We recommend using as small a gate as possible (too small of a gate will over-shear the material). A gate size no smaller than 0.5 mm (0.020") and no larger than 3.0 mm (0.120") is recommended. Refer to the Quick Processing References on our website for more information on gating using specific grades of our TPVs.

Restricted gate
A restricted gate slows the flow of material around the gate area, which increases the shear of our TPV materials. Restrictions could be caused by a rapid wall thickness change in the part. We recommend wall thickness transitions of 15 percent or less. Burrs or sharp corners at the gate will also reduce flow at the gate. Polishing the gate area can reduce this restriction.

Injection speed too high
Most of our TPVs process using fast injection rates, achieving fill times ranging from 0.5 to 1.5 seconds. However, trying to inject too rapidly will increase the shear rate of melt flow in the gate.

Back pressure too high
A rapid withdraw of the screw will over-shear the material. The last place for melt flow to set up is around the gate, which increases the chance of a burn mark in this location.

Mold temperature at gate too high
Reducing the temperatures around the gate will reduce burning at the gate.

Front barrel zone temperature too high
Reducing front barrel zone temperatures will reduce burning at the gate.

PROBLEM: BURN SPOTS ON PART

Injection rate too high
When TPV material is injected too fast, trapped air compresses and ignites. By reducing injection speeds, air will have more time to escape. Traditionally, injection speeds should be set to achieve a fill time of 0.5 to 1.5 seconds.

Screw RPM too high
We recommend a high screw RPM (100 to 200) to ensure high shear during plasticizing. However, when screw RPM is too high, there may be an over-shearing of the TPV material. This can cause scorching of the material.

Back pressure too high
We recommend using only enough back pressure needed to provide good mixing of TPV material. The minimum back pressure recommended is 0.3 MPa (50 psi). Higher back pressure could over-shear TPV materials and might even mix in air which could ignite and cause a burn mark towards the last place to fill.

Clamp pressure too high
Excessive clamping force can shut off or crush vents. The machine should be capable of applying a clamping force of 4.0 to 6.9 kN/cm2 (3 to 5 tons/in^2) of the projected area.

Venting
Generous venting is necessary when using our TPVs because of their fast filling times. As molten TPV material enters a mold quickly and under high pressure, enclosed air must exit just as quickly. Burn marks located at the end of fill are usually a sign of improper venting and are caused when trapped air ignites (called dieseling).

Venting recommendations for our TPVs:

TPV family	Rec. max. depth	Land length	Width
General purpose grades	0.038 mm (0.0015")	3.8 to 5.0 mm (0.15 to 0.20")	5.0 to 12.0 mm (0.20 to 0.50")
Molding grades	0.025 mm (0.0010")	3.8 to 5.0 mm (0.15 to 0.20")	5.0 to 12.0 mm (0.20 to 0.50")
High flow grades	0.025 mm (0.0010")	3.8 to 5.0 mm (0.15 to 0.20")	5.0 to 12.0 mm (0.20 to 0.50")

TROUBLESHOOTING

PROBLEM: DISCOLORED STREAKS OR CHUNKS

Incompatible color or additive carriers
We recommend using only polyolefin carriers, such as polypropylene or polyethylene, for colorants and additives. Never use polyvinyl chloride-based additives because they can react with TPVs at typical processing temperatures and cause a safety hazard. To rule out an incompatible color or additive carrier as a cause, run natural virgin material under the same conditions. If the problem disappears, it is likely associated with an additive.

Prior runs have contaminated the plastification system
We recommend always physically cleaning the entire injection molding system, including the hopper, barrel, screw, shut off valve, nozzle and mold, prior to beginning a run with our TPV materials. Foreign materials in these areas may not blend properly with our materials and cause streaking or chunks in the part. The system can be purged using polyethylene or polypropylene. Never use polyvinyl chloride or its derivatives to purge the system.

Insufficient blending of color concentrate
Blending of our TPV materials and the color concentrate happens when both materials reach the melt phase. Check our Injection Molding Guide to ensure that melt temperature is within the recommended processing range. If melt temperature is correct, then focus should be placed on increasing the shear, heating and mixing of material in the barrel.

- First try increasing the back pressure
- Next try reducing the rear zone temperature
- If these suggestions do not work, it may be necessary to replace the screw or nozzle with one that has a mixing design

PROBLEM: DISTORTED PARTS (WARPING)

Stresses and material orientation are leading causes of distorted parts. The finished part will appear to be warped in some fashion. Non-uniform shrinkage caused by stresses in the part traditionally causes warping of a part. The source of the stresses in a part is very complex, and often a combination of many forces.

Molded part stresses
Material will generally shrink toward the hotter areas of the melt. If there is a large temperature differential existing in the melt, stresses will be created in the part. To address this issue, increase the mold temperature to increase the cooling time. Prevent over-packing at the gate by reducing hold pressure or hold time. Also, increase the melt temperature and delay ejection to ensure the part has sufficiently cooled and solidified prior to ejection.

Difference in packing density
If there are non-uniform pressures in the mold during the injection and pack/hold phases, the part will not have uniform densities. Increase the injection fill speed to reduce material viscosity as it enters the cavity. Increase screw RPM to ensure a more homogeneous melt and reduce viscosity of flow. Also, increase the hold pressure. If there is a sharp variation in cross-sectional thickness, this could lead to pressure variations in the part, which will also affect the packing density.

Ejection is operating improperly
If ejector pins are actuated too early, are sized too small or are located improperly, they may be deforming the part. First, ensure that the ejector pins are moving in a uniform manner across the part. Ejector pins should be located in thicker, stronger areas of the part. If the bearing area of the ejector pins is too small, they can damage the part. Increase the diameter and number of ejector pins to spread the ejection pressures more uniformly over the surface of the part. With delicate parts it may be necessary to use air instead of pins to eject the part.

PROBLEM: FLASH ON PARTING LINE

There are a few key reasons why flash appears on the parting line of a part. First, excessive injection pressures may be forcing the mold apart. Ensure that injection pressures are appropriate (under 70 MPa [10,000 psi]). Check that the machine is creating enough clamping force to keep the mold halves together. We recommend the machine be able to apply forces between 4.0 and 6.9 kN/cm^2 (3 and 5 tons/in^2) of the projected area. Finally, the mold may be improperly aligned, damaged, worn or have over-sized vents.

Excessive injection pressure causes mold to part
Flash will occur during fill when the injection pressures are greater than the clamping force. Using injection pressures meant for a lower viscosity material can flash the mold. It is important to note that a distinction must be made whether the flash is occurring during fill or during the pack and hold stage. This can be determined by doing a fill-only shot. Adjust the pack and hold pressures to zero and use velocity only to fill the mold (95%). If the part is flashing at this point, the problem is occurring during the fill. Decreasing injection speed, verifying injection fill time and moving the transfer point back are steps that can help alleviate this problem. Finally, increase pack and hold pressure to 50% of the injection pressure setting and pack and hold times to 2 to 3 seconds to ensure that the part is filled. Increase pack and hold time until the part weight does not change (refer to Start Up Procedures).

Melt temperature is too high
Material viscosity can also cause flash. Temperature can affect the viscosity of our TPV materials, though not as significantly as shear. When temperatures are too high, viscosities may be too low. To increase viscosity, systematically reduce the temperature settings of the nozzle, front barrel zone and mold in 5 to 10° increments. Finally, if no reduction in flash is observed, reduce the screw RPM by 5 to 10% to decrease the shear, which in turn will increase the viscosity of the melt.

Mold function is incorrect
The last, and many times the most expensive, remedy involves the mold. Before reworking the mold, check to make sure mold faces and machine platens are aligned properly. Make sure there is no foreign matter on the mold faces. We do not recommend using any release agent on the mold surface. Check for damage on the mold, wear or improper venting. For proper venting instruction, refer to the venting section in the Injection Molding Guide.

PROBLEM: JETTING

Jetting occurs when the melt flow exiting the gate or other restrictive area enters a mold cavity and does not come in contact with the mold wall or some type of impingement. The abrupt drop in pressure creates a disorganized, higher viscosity flow into the open cavity. Snake-like tracks will appear across a part. This effect is actually a result of multiple miniature weld lines where the melt folds over itself. Apart from a poor finish, it will also create part weakness.

Mold design
A gate design that encourages melt flow to travel towards a wall or a pin instead of into a free, unrestricted area is preferred.

- Leave a maximum impingement distance of 6.35 mm (0.25") between a gate and the opposing wall
- If this is not possible, changing the angle of the gate to increase the impingement of the melt flow along the mold wall may be an option
- Installing a pin just before the gate or a spring-loaded pin just after the gate may help to disperse material along the mold wall

It is, however, always recommended to have a gate location that maximizes melt-to-mold contact in the beginning. It is possible that jetting will only be reduced by changing the location of the gate.

TROUBLESHOOTING

PROBLEM: MATERIAL DROOL FROM NOZZLE

Nozzle drool is usually a sign of a substantial reduction in the viscosity of the material. A primary cause of the reduction in viscosity is material degradation. Nozzle drool can also be caused by built up pressure in the barrel and excessive heat in the front zone or nozzle.

Material degradation
Material degradation can cause a drop in melt viscosity which, under pressure, will leak from the nozzle. Be sure that material is sufficiently dried. Since most of our TPVs are slightly hygroscopic, drying the material at 82°C (180°F) for at least 3 hours (at least 4 hours for blends with regrind) is recommended. Reduce residence time or melt temperature to ensure that material is not being degraded in the barrel. If the machine is left idle at processing temperatures for more than 20 minutes, it is good practice to purge the material inside the barrel.

Pressure build up
Using decompression to deliver a short burst or pull back of the screw after it stops turning so the material is pulled under vacuum may eliminate the drool. Use caution when using decompression to eliminate drool. Excessive decompression can cause too much air to be sucked in through the nozzle.

Barrel and nozzle temperature
Check our Injection Molding Guide for correct processing temperatures. Another cause of drool may be improper gate selection when using hot runners (see the hot runner section of our Injection Molding Guide).

PROBLEM: PART DIMENSIONS TOO LARGE

Larger than expected dimensions in a part can be caused by increased pressures in the mold. 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. Pressures in the mold work to increase the contact between the shot material and the mold surface. Generally, high injection pressures maintain part properties, improve part appearance, and reduce molding cycle times. However, too much pressure during the fill, pack, and hold cycle could lead to an over-packing of material in the mold. Verify that the set up conditions are correct using the Start Up Procedures.

Mold is over-packed
Mold over-packing is a leading cause of oversized parts. Over-packing, however, will do more than cause larger parts; it will increase processing and maintenance costs. The parts will consume an excessive amount of material, increase rejection rates (flashed parts), and cause damage to the parting line.

- First, verify that injection fill time is correct. Typically, mold fill time takes between 0.5 and 2 seconds, depending on part volume, runner-gate style and size, cavity locations and injection pressure.
- Next, reduce the hold pressure, pack pressure and/or pack time in these instances to help alleviate the problem. We also recommend a small cushion of 3.2 to 6.35 mm (0.125 to 0.250"), especially with softer grades that can com press more during the injection and packing phase.

Mold designed incorrectly
We always recommend using a "steel safe" mold for prototyping prior to building a production tool. If the dimensions of the mold are too large, steel addition may be an option.

PROBLEM: PART DIMENSIONS TOO SMALL

Smaller than expected dimensions in a part can be caused by decreased pressures in the mold. 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. Pressures in the mold work to increase the contact between the shot material and the mold surface. Generally, high injection pressures maintain part properties, improve part appearance, and reduce molding cycle times. A reduction of pressure in the mold cavity can be caused by the gate freezing off prematurely, poor mold design, or low pack/hold pressures. Verify that the set up conditions are correct using the Start Up Procedures.

Mold is under-packed
Mold under-packing causes poor contact between the shot material and the mold walls, which seriously reduces cooling efficiency. Under-packing of the mold cavity can lead to excessive shrinkage, shrink voids in thicker wall sections, sinks in large surface areas, and poor part appearance. Typically, mold fill time takes between 0.5 and 2 seconds, depending on part volume, runner-gate style and size, cavity locations, and injection pressure. We also recommend reducing the cushion in the cylinder to between 6.0 and 3.2 mm (0.250 and 0.125"), especially with softer grades that can compress more during the injection and packing phase.

If these suggestions do not work, it may be necessary to decrease the viscosity of the material entering the mold cavity. Increasing the injection velocity, back pressure, and screw RPM will increase shear (lowering viscosity) of the TPV material and ensure a more homogeneous mix of the melt. A lower viscosity, more homogeneous melt will improve packing out of the cavity. Our TPV materials depend mainly on shear to increase flow. However, increasing melt temperature will also lower the viscosity of our materials to a lesser extent, and is an option.

Mold temperature is too high
Ensure that mold temperatures are not too high. Refer to the Quick Processing References on our website for information on specific grades of our TPVs.

Mold designed incorrectly
We always recommend using a "steel safe" mold for prototyping prior to building a production tool. If the dimensions are too small, removal of steel in these areas will increase part size.

PROBLEM: PART STICKS IN MOLD CAVITY OR SPRUE

Pressures in the mold work to increase the contact between the shot material and the mold surface. Generally, high injection pressures maintain part properties, improve part appearance and reduce molding cycle times. However, if there is too much pressure in the cavity, it can cause the material to stick to the mold wall. Also, as parts cool and solidify they shrink away from the mold walls. If the part is not sufficiently cooled, then the part can stick in the mold. Other causes of this problem are associated with mold and part design.

Part is over-packed
Over-packing can lead to parts sticking in the mold. Over-packing, however, will do more than cause sticking parts; it will increase processing and maintenance costs. The parts will consume an excessive amount of material, increase rejection rates (flashed parts) and cause damage to the parting line. Please refer to the Start Up Procedures for the injection molding process.

Part is not sufficiently cooled
If there is not enough time to cool the part to a point where it can pull away from the mold wall, it could stick to the mold cavity. Increasing cooling time will help. Reducing melt and mold temperatures will decrease the time needed to cool the part.

Mold design is incorrect
The part may be sticking in the mold because either the sprue is sticking or due to mold design and polish. If the sprue is sticking in the tooling, check that the nozzle is correctly aligned and sealed with the sprue bushing. If it is not aligned properly, it can cause material seepage, which will act like an undercut and hold the material in the cavity. If this is not the case, try reducing the length of the sprue to decrease the surface area/hold that the sprue cavity has on the material. If the part is sticking to the mold surface, it could be that the part/mold design is incorrect.

Tool steel
We recommend having a minimum draft angle of 0.5° on mold walls to reduce surface area drag of ejecting parts on the mold wall. We also recommend using an SPI/SPE-B #3, #4, #5 or #6 finish (VDI 30-36 / Ra 3-6) for good mold release. For higher polished surfaces, air assisted ejection is required for easy part removal and minimum distortion. We suggest a minimum draft of 0.5° per each 0.025 mm (0.001") depth of texture used. When polishing the mold, always draw polish. Polishing in the direction the part will be ejected will reduce the amount of small ridges that can hold back the part. Finally, make sure that any cores in the mold are strong enough to handle the injection pressure. If the core bends slightly from injection pressure, it will create an undercut in the part and hold the part in the mold. Thickening core areas and using stronger steel in these areas is recommended.

TROUBLESHOOTING

PROBLEM: POOR FINISH

When molding Santoprene TPVs and our other TPE products, pressures in the mold work to increase the contact between the shot material and the mold surface. Generally, high injection pressures maintain part properties, improve part appearance and reduce molding cycle times. When the melt flow pressures do not allow for the material to pack out evenly against the mold cavity, poor surface finish can result.

Mold fill is too slow
The majority of our TPVs are shear dependent. They require fast mold filling via high injection speeds and pressure to increase shear through the system, which reduces their viscosity. If increasing injection pressure and injection speed does not remedy a poor finish, increasing the melt temperature and mold temperature will also improve surface appearance when molding our materials.

Mold function is incorrect
First make sure that venting and gate size are not reducing the melt flow speed. Vents that are too small can lead to trapped air in the mold cavity, which can slow melt flow into the cavity. We do not recommend using mold releases or lubricants. Over time, the lubricants may actually cause the parts to stick in the mold due to build up and vent clogging. Be sure that the mold surface is clean. Avoid thick ribs and bosses that will create sink marks on the opposite side of the part.

PROBLEM: POOR SURFACE DETAIL

The leading causes of surface detail problems on parts molded from Santoprene TPVs are improper filling and packing conditions. Pressures in the mold work to increase the contact between the shot material and the mold surface. Generally, high injection pressures maintain part properties, improve part appearance, and reduce molding cycle times. A reduction of pressure in the mold cavity can be caused by the gate freezing off prematurely, poor mold design, or low pack/hold pressures. Verify that the set up conditions are correct using the Start Up Procedures.

Mold filling improperly
If material is not filling surface details on the part, there may not be enough pressure or material to fill the part. First check that the shot size is correct. If the shot size is correct, try increasing the injection speed and back pressure. This will help ensure there is enough pressure in the mold to encourage contact between the melt flow and the mold surface.

Packing conditions are improper
Under-packing of the mold cavity can lead to excessive shrinkage, shrink voids in thicker wall sections, sinks in large surface areas, and poor part appearance. Traditionally, mold fill time takes between 0.5 and 2 seconds, depending on part volume, runner-gate style and size, cavity locations, and injection pressure. Increasing the second stage pressure and time will also help to maintain pressures in the mold necessary for good surface finish.

Polymer cooling rate is too high
Increasing mold temperatures will help by reducing the cooling rate of the melt and encouraging filling of surface details.

PROBLEM: POOR WELDS ON PART

Weld marks on a part are caused when two melt flow fronts come together. If the melt fronts have distinctly different temperatures or are too cool, they will form a line on the part where the materials come together. In some cases weld lines can cause part weakness, along with being aesthetically displeasing. Gating correctly to minimize weld lines and promote efficient flow is always suggested. To prevent weak and apparent weld lines, ensure that when the flow fronts reach each other, they have a low enough viscosity and sufficient pressure behind them to allow proper bonding. Hold times may need to be extended to allow for solidification of the bonded flow fronts. Finally, as flow fronts come together they will tend to trap air between them. Sufficient venting is recommended in these areas.

Polymer cooling rate is too high
When two cool material fronts come together, a weld line will appear. Increasing melt and mold temperatures will help maintain heat and lower viscosity in the flow fronts. It is also important to make sure there is enough pressure behind the flow fronts to ensure a sufficient bond. Increasing injection pressure and injection speed will help increase pressure in the mold cavity. Lengthening cycle time will encourage solidification of the bonded flow fronts.

Mold design and/or function are improper
The location and size of the gate are important to maintain efficient melt flow. The location of the gate should minimize the number of flow fronts in the mold if possible. The gate should be small enough to ensure proper shear, yet large enough to allow proper flow. We recommend a gate size no smaller than 0.5 mm (0.020") and no larger than 3.0 mm (0.120"). Refer to the Quick Processing References for details on gating using specific grades of our TPVs.

Generous venting is necessary when using our TPVs because of their fast filling times. As molten TPV material enters a mold quickly and under high pressure, enclosed air must exit just as quickly. For large parts or parts with long flow lengths, it may be necessary to reduce the fill speed to allow time for air to escape the part. Increasing the mold temperature will also help material flow by maintaining the melt temperature and reducing the cooling rate of the material.

TROUBLESHOOTING

PROBLEM: SHORT SHOTS PREVENT COMPLETE MOLD FILLING

Short shots are caused when there is not enough material entering the mold cavity during the fill process. The problem may be under-packing (also see "Part Dimensions Too Small") of the mold. The conditions for correcting dimensional shortages are similar to remedying short shot problems. Simply, there needs to be enough melt flow with enough pressure and proper fill and hold time to allow for filling the cavity. An incomplete part will be produced with a short shot, which distinguishes it from dimensional problems.

Available melt volume is insufficient
Increasing shot size is the obvious solution to increasing the amount of the material in the mold cavity. However, we suggest a cushion of 6.35 to 3.2 mm (0.250 to 0.125") to adjust for variations in different screws and nozzles. If a short shot still exists, increasing first stage injection speed or velocity will increase shear in the melt flow and lower viscosity, requiring less pressure to fill the cavity. Increasing nozzle and front barrel temperature will also decrease viscosity by adding more heat to the melt flow prior to it entering the mold. If these suggestions do not solve the problem, there may be an issue with the physical equipment restricting the melt flow. Ensure that nozzle and runners are clear of obstruction and that the non-return check valve on the screw is operating properly.

Mold has high pressure drop
A reduction of pressure in the mold cavity can be caused by the gate freezing off prematurely, poor mold design, or low pack/hold pressures. We always recommend gating a part in the thickest section with flow towards thinner areas. This helps ensure that pressures in the mold are balanced and maintained. If a large cavity is receiving melt flow from a narrower channel, there will be a substantial drop in the pressure pushing the melt flow. It can cause the narrow channel to freeze off, denying the large cavity an opportunity to fill. Moving or adding a gate to the thicker area may alleviate this problem. However, care must be taken when adding a gate as it will create another flow front in the mold and may require additional venting where the flow fronts meet. If this is not an option, maximizing the pressure in the mold and reducing the viscosity of the material may be the only options. Increasing the back pressure and injection speed will increase shear (lowering viscosity) of the TPV material and ensure a more homogeneous mix of the melt. Increasing pack/hold pressures will increase the pressure in the mold, helping the melt flow pack out. If these suggestions do not solve the problem, it may be necessary to use a higher flow grade of our TPVs.

Mold function is incorrect
Restrictions in a mold cavity or an unbalanced multi-cavity tool can also cause short shots. Be sure that all gates in a multi-cavity tool are the same size. For balanced melt flow and cavity filling, design runners for the shortest, straightest flow path consistent with the best gate location(s). In order to reduce pressure drop and facilitate heat retention, the ratio of surface area to volume should be kept to a minimum. Runners also function to transfer pressure and regulate the pressure drop between an injection molding machine and a mold cavity. Also, ensure that the mold is vented correctly and that vents are clear, allowing air to escape.

PROBLEM: SILVER STREAKING

Silver streaking occurs when moisture is introduced into the system or when there is degradation of the melt flow. If colorant is added, we recommend using only polyolefin carriers for colorants and additives, such as polypropylene or polyethylene. Never use polyvinyl chloride-based additives because they can react at processing temperatures typical with TPVs and cause a safety hazard. To rule out an incompatible color or additive carrier as a cause, run natural virgin material under the same conditions. More information on color compatibility can be found in the "Incompatible Color or Additive Carriers" section of our Injection Molding Guide.

Material has trapped moisture
Most of our TPVs are hygroscopic. Effective moisture control ensures high quality parts that have an appealing surface appearance. With any TPV, moisture impacts processing, physical properties, product performance, appearance or a combination of these properties, along with silver streaking. We recommend drying our TPVs before processing them using a desiccant drying system. Hot air systems are not recommended, and can actually increase the moisture level of TPVs in high humidity environments. Typically, our materials are dried at 82°C (180°F) for at least 3 hours (at least 4 hours for blends with regrind). More recommendations on drying can be found in our Injection Molding Guide. Improper venting can also trap moisture in the part. Humidity in the material that is unable to vent off or in air trapped in the mold will cause silver streaking.

Melt temperature is too high
Over-shearing or over-heating of our TPV materials will cause degradation. We recommend a high screw RPM (100 to 200) to ensure high shear during plasticizing. However, when screw RPM is too high, there may be an over-shearing of the TPV material which will cause scorching. Reducing barrel set point temperatures will also alleviate decomposition of our materials.

Mold condensation
Over-cooling a mold will produce condensation on the mold cavity walls. When melt flow moves across the cavity wall, it picks up the moisture and causes silver streaking. Increasing mold temperatures will help dry mold cavities and should reduce streaking.

PROBLEM: SINK MARKS

Sink marks are dimple indentations on the surface of a part. Part design, mold function, and processing parameters can all contribute to conditions that cause sink marks (also see Part Dimensions Too Small).

Poor part design
Thick wall sections relative to the rest of the part will cause a sink mark on the opposite side of the part. A wall thickness greater than 6.35 mm (0.250") will lead to sinkmarks. Bosses and ribs are especially susceptible to creating sink marks. Keeping wall thickness as constant as possible and coring thicker areas is recommended.

Mold function is incorrect
First make sure that venting and gate size are not reducing the melt flow in the cavity. A gate that is too small will restrict injection speed and vents that are too small can lead to trapped air, slowing melt flow into the cavity. Reducing the length of sprue bushings and increasing the bushing diameters will also help maintain melt flow pressure and temperature entering the mold cavity. Also, ensure that mold temperatures are not too high. A mold that is too hot, especially with semi-crystalline materials like our TPVs, will cause more shrinkage. Increasing cooling time, especially for thicker parts, will reduce the shrinkage causing the sink mark.

Processing parameters
Mold under-packing causes poor contact between the shot material and the mold walls, which reduces cooling efficiency. Under-packing of the mold cavity can lead to excessive shrinkage, shrink voids in thicker wall sections, sinks in large surface areas, and poor part appearance (see Start Up Procedures). If these suggestions do not work, it may be necessary to decrease the viscosity of the material entering the mold cavity. A lower viscosity, more homogeneous melt will improve packing out of the cavity. Our TPV materials depend mainly on shear to increase flow. However, increasing melt temperatures will also lower the viscosity of our materials to a lesser extent, and is an option.

TROUBLESHOOTING

PROBLEM: SPLAY MARKS IN GATE AREA

Generally, high injection pressures maintain part properties, improve part appearance, and reduce molding cycle times. A reduction of pressure in the mold cavity can be caused by the gate freezing off prematurely, poor mold design, or low pack/hold pressures. Since the gate area is the last place where molten material solidifies, this is the location splay marks or blemishes will show up if pressures are not maintained in the mold. Verify that the set up conditions are correct using the Start Up Procedures.

Low pressurization at gate area
Pressures in the mold work to increase the contact between the shot material and the mold surface. A localized area of low pressure at the gate will cause poor surface detail at the gate. First, increase velocity and pack/hold time to help ensure there is enough pressure to pack out the mold. Next, increase the back pressure and screw RPM to increase shear (lowering viscosity) of the TPV material and ensure a more homogeneous mix of the melt. Increasing the mold temperature will reduce the cooling rate of the material and ensure a good surface finish. Finally, reduce decompression. There may be air being sucked back into the melt flow which will show up in the next shot, affecting surface area and causing voids in the part.

Mold set up is incorrect
Make sure the sprue and runner system are directed and sized correctly to ensure fast, efficient flow. Runners regulate the melt flow from the sprue to the cavities. For balanced melt flow and cavity filling, design runners for the shortest, straightest flow path consistent with best gate location(s). In order to reduce pressure drop and facilitate heat retention, the ratio of surface area to volume should be kept to a minimum. Runners also function to transfer pressure and regulate the pressure drop between an injection molding machine and a mold cavity. Shortening the land length of the gate will also help regulate pressure and aid in heat retention.

PROBLEM: VOIDS IN MOLDED PART

There are two reasons for voids appearing in a part:
- Material in the core solidifying slower than material on the surface
- Moisture in the melt flow

Material in core sets up slower than surface material
Melt flow cools quickly against cool mold surfaces. It takes longer for the center of the part to cool. This is especially true for parts with very thick cross-sections. Melt flow will continue to shrink until it solidifies. The difference in the shrinkage between the center and the surface creates voids in the part. Increasing pressure in the mold will force material towards the mold wall until the material solidifies, reducing the gaps. To increase the pressure in the mold, increase injection speed, hold pressure, and hold time. Make sure that the sprue, runner, and gates are large enough to assure free melt flow. Increasing the mold temperature and cooling time will slow down the solidification of the material against the surface of the mold. Increasing the back pressure and screw RPM will increase shear (lowering viscosity) of the TPV material and ensure a more homogeneous mix of the melt. A lower viscosity, more homogeneous melt will improve packing out of the cavity.

Moisture in the pellets
Most of our TPVs are slightly hygroscopic. Effective moisture control ensures high quality parts that have an appealing surface appearance. With any TPV, moisture impacts processing, physical properties, product performance, appearance or a combination of these properties, along with silver streaking. When moisture is trapped inside a part, it will create a void. Improper venting can also trap moisture in the part. It is important to use sufficient venting when using our TPVs. See the drying section of our Injection Molding Guide for more details.

Energy lives here™

Contact us for more information:
exxonmobilchemical.com

S0817-040E49