Advanced Composites

# PP/TPO Processing Guidelines and Troubleshooting Guide

This guideline provides valuable information to help with some of the many problems that may arise when working with polypropylene. The suggestions and material supplied should be considered as general guidelines. Each part has to be designed with its end use requirements in mind. If additional information is needed, please contact a process engineer.

6/1/2007

 Advanced Composites TS Group

# Table of Contents



 

Introduction:
Advanced Composites (ACP) is a world class PP/TPO compound supplier to the Automotive Industry. These materials meet the ever-demanding needs of the automotive interior and exterior communities.

This guideline will give you an understanding of the possibilities and requirements of ACP materials. Baseline processing parameters and design guidelines will also be provided.

Please utilize our Technical Service Department for more detailed information.

## PP/TPO Characteristics

High Temperature Resistance
Polypropylenes relatively high melting point of 335°F allows continued use at temperatures up to 220°F with the material softening at 235°F. We have continued success with the molding of instrument panels for the automotive field, which has very aggressive heat and sun load testing requirements.

Low Density
The density of Polypropylene compounds can range anywhere from 0.90-1.15 g/cm$^3$. This low density allows more parts to be made per pound compared to a higher density engineering type resin.

UV Resistance
Polypropylenes and the polypropylene compounds can be formulated to have excellent UV Resistance. This allows molded parts to be exposed to sunlight for long periods with little or no deterioration in appearance and properties. ACP compounded resins to be used for interior as well as exterior applications without the use of paint.

Chemical Resistance
The Polyolefin family of thermoplastics is highly resistant to most inorganic chemicals with few exceptions. Non-polar organic compounds (toluene, benzene, etc.) could cause some swelling and softening of the material.

Other advantages
ACP has the knowledge and ability to meet the increasing challenges of the Automotive Industry. We continuously work with our customers to supply them

the material they need at the right price. We offer excellent technical support to ensure that your design and processing needs are met and exceeded.

## Part Design

Wall Thickness

When designing a part to be molded from PP/TPO compounds it must be understood that a wall thickness of 2.5-3.5mm is optimal. Whenever possible there should be no filling from thin areas to thicker areas. For best molding conditions, it is imperative that the nominal wall be kept consistent.

Ribs

The use of ribs is very common with polypropylene in order to use a thinner wall thickness to achieve the same overall stiffness and strength. It is very important that the design of the rib be correct to eliminate read through and sinks. The design of the rib should be as shown in Figure1.

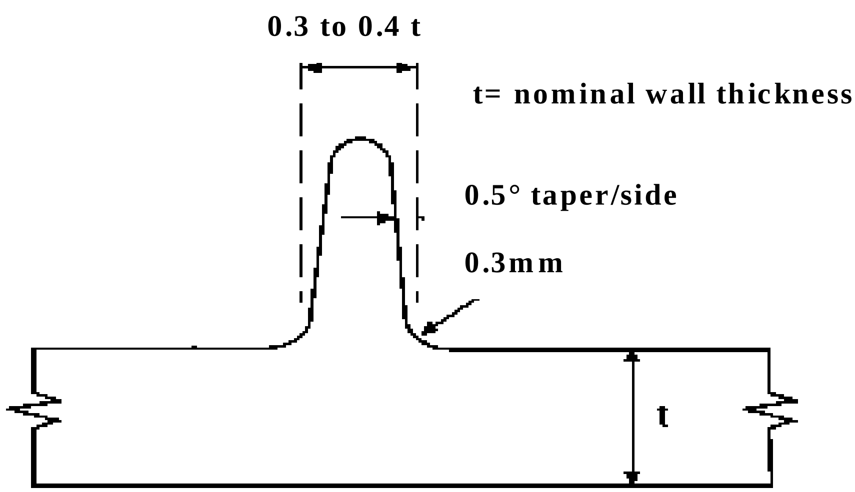


*Figure 1: Standard Rib Design*

The rib base (closest to the wall) should be approximately 30% of the nominal wall, with a radius of 0.3mm. The rib should be drafted (0.5°) to allow for easy ejection.

Radii

Whenever possible, inside and outside corners should have radii to limit the notch sensitivity of polypropylene and TPO.

Undercuts

Undercuts can be molded with the use of lifters or slides. It should be noted that a relatively highly polished surface is required in order to allow the material to release itself from the lifter. Burrs are a very big problem with the use of lifters and should be removed before use.

Holes

Holes can be successfully molded in a PP/TPO part. It must be kept in mind that a knit line will form 180°on the opposite side from the gate location.

Draft

Failure to provide adequate draft may result in parts sticking in the tool and/or grain drag. Part design will dictate required draft angles. For Class A grained surfaces, the texture source will dictate the required draft.

Living Hinge

Living hinges can successfully be molded with PP/TPO, please consult a Technical Representative from ACP for details on design.

## Mold Design

Manifold Type

The use of a hot runner manifold is common when molding large parts or parts with multiple gates. Use the following as guidelines for selecting the proper manifold diameter.

| PART DEFINITION | VERY SMALL | SMALL | MID-SIZE | LARGE | EXTRA LARGE |
|---|---|---|---|---|---|
| Manifold | N/A | 0.625" – 0.75" Diameter | 0.625" – 0.75" Diameter | 0.75" – 0.875" Diameter | 1.0" Diameter |

When using a manifold use the following as a guideline for the nozzle diameter. Keep in mind that a slightly larger diameter may be needed for the use of Valve Gates.

| PART DEFINITION | VERY SMALL | SMALL | MID-SIZE | LARGE | EXTRA LARGE |
|---|---|---|---|---|---|
| Nozzle | N/A | 0.5" – 0.625" Diameter | 0.5" – 0.625" Diameter | 0.625" – 0.75" Diameter | 0.75" – 0.875" Diameter |

## Gate geometry

There are many different types of gates that can be used in injection molding. Below is an example of some common gate types.

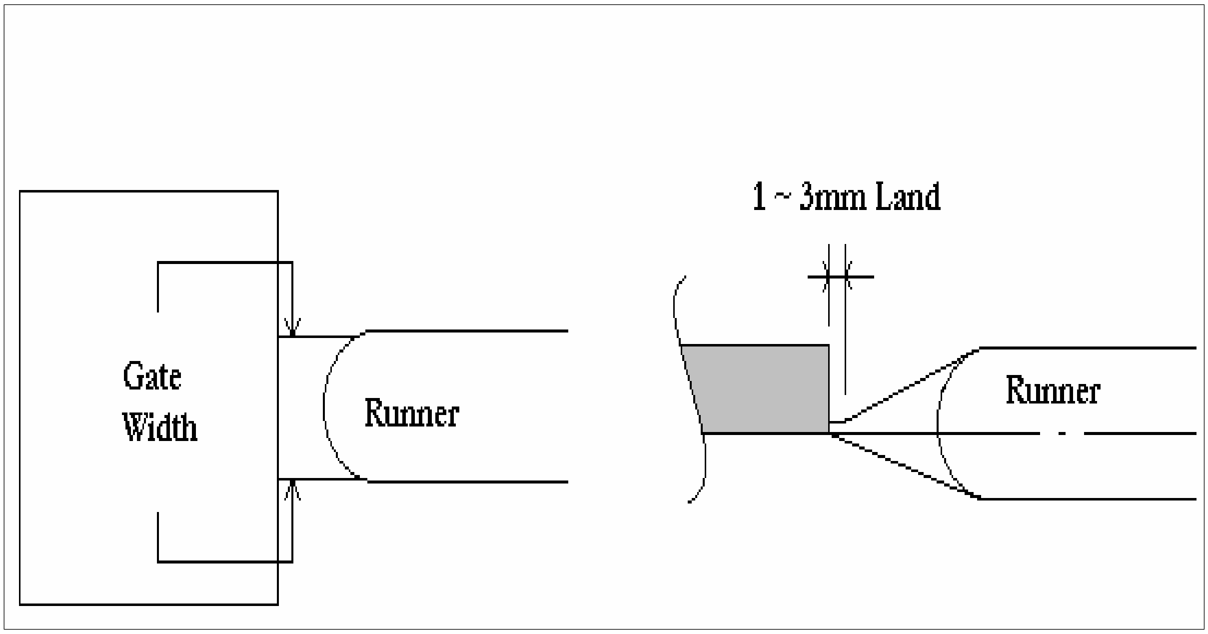


*Figure 2: Edge Gate (example)*

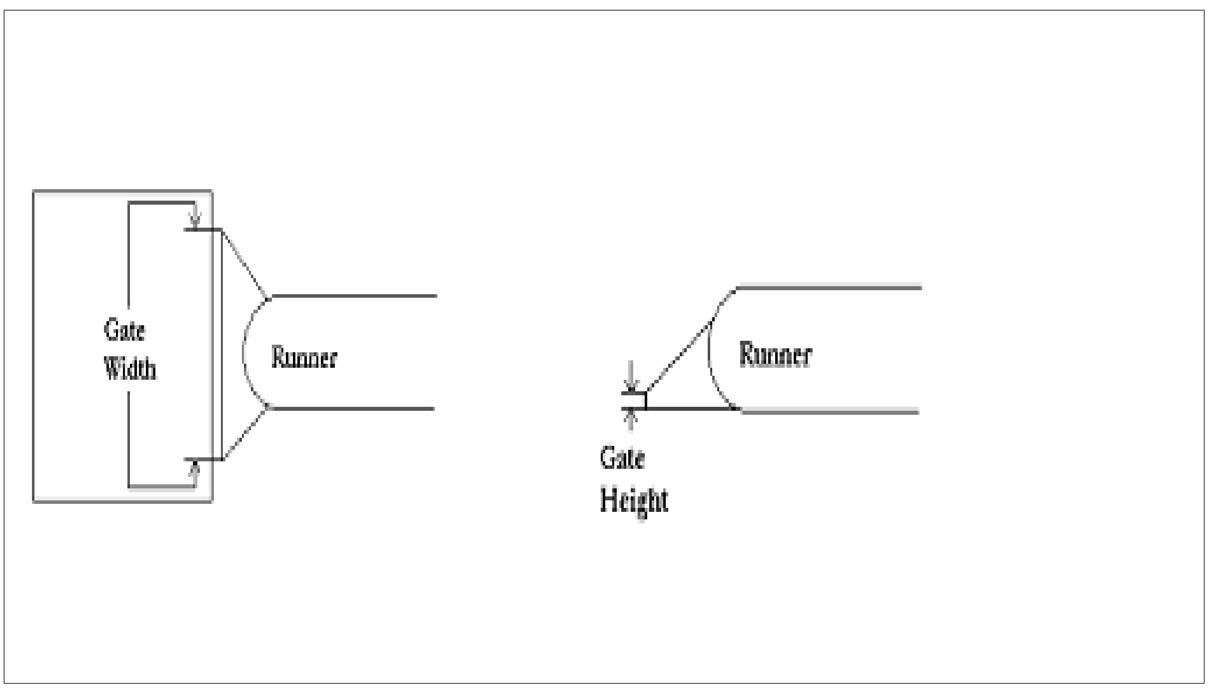


*Figure 3: Fan Gate (example)*

 

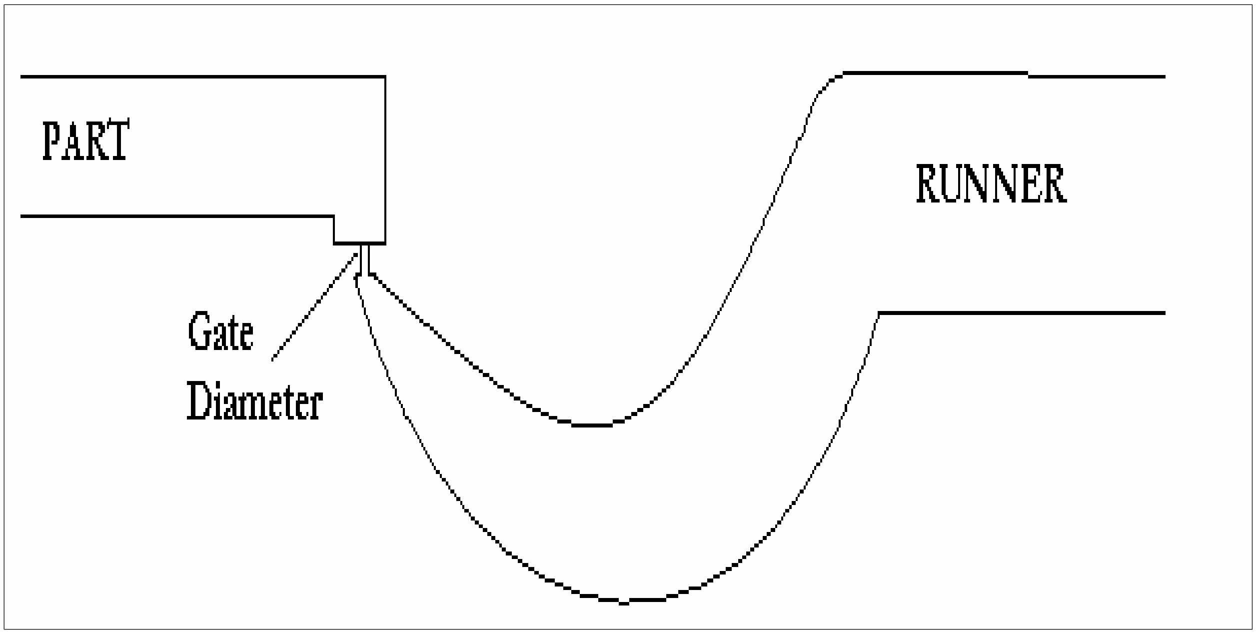


*Figure 4: Cashew Gate (example)*

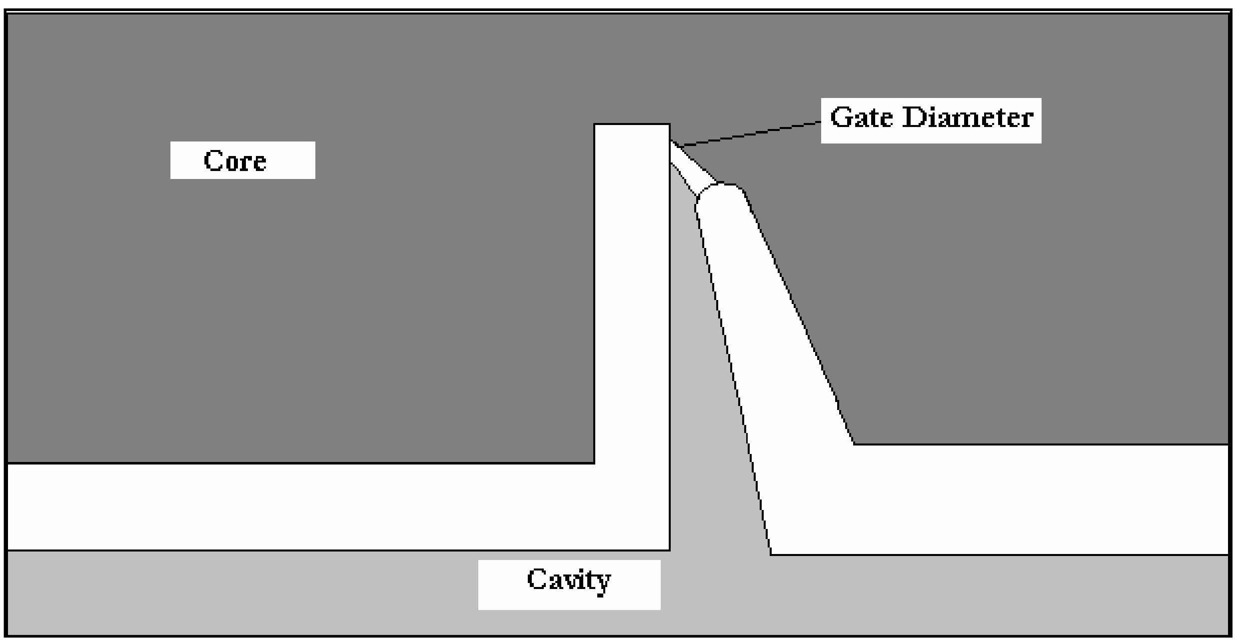


*Figure 5: Tunnel (submarine) gate (example)*

 

Gate Location

A mold filling simulation software should be used to determine gate location. The software can be used to determine the best gate location to eliminate weld lines, evenly distribute pressure in the cavity, keep overall melt front temperature similar, and predict generic processing conditions. Advanced Composites can help with this process if necessary. Please contact us if you need Moldflow data.

Venting

Proper venting is required in all injection molds. When using a PP/TPO Compound it is very critical to have proper vent depth and location. A vent depth of 0.0015" is typical and location can vary from part to part. The end of fill area along the parting line should have more vents than the rest of the parting line, but all areas need to be considered. Venting of ejector pins and lifters is also used in many applications. Common defects from insufficient venting are bubbles, splay, burning, and high gloss.

Mold Cooling Requirements

Mold cooling is one of the most important variables in injection molding. Proper mold cooling can eliminate warp, decrease cycle times, and increase part quality. Mold filling simulation software should be used to ensure that water line location is correct. Whenever possible, water manifolds should not be used, as this will not allow proper flow to each water line. Water leaving a water channel should not be more than 3-5°F different from the water going into the tool. If there is a large $\Delta T$ then there is not enough flow through the cooling lines. Any restrictions should be eliminated to allow water to flow freely.

Cores (Holes)

When possible all edges of the hole should have a radius to help promote flow and the center of the core should be vented to atmosphere.

Part Ejection Requirements

When molding PP/TPO Compounds, it is critical to have proper ejection, whether in the form of ejector pins, lifters, or stripper plates. All types of ejection should have some venting that will allow gases to escape into the wall between the pin and core. Ejector pins should be cleaned off regularly and should have minimal grease on them, this will eliminate contamination on the final molded part.

 

## Molding Parts with PP/TPO Compounds

Machine Specifications

When molding PP/TPO compounds certain equipment is needed.

1. A properly sized gravimetric blender is preferred when mixing natural material and concentrate. This may be done at the press or in a central location.
2. If using Sequential Valve Gates, ACP recommends triggering on screw position.
3. When using our materials an aggressive mixing screw may be needed. A distributive mixing screw with a high shear spiral mixing section on the front is optimal. This combination will ensure that the material is properly mixed before injecting it into the mold. ACP will work in conjunction with a screw supplier to help design the appropriate screw.
4. ACP strongly recommends that our materials be dried. A desiccant drier is preferred. See the appropriate material processing sheet for recommended conditions.

Start Up Molding Conditions

Please refer to the corresponding processing sheet for the material that you will be using to find the appropriate processing information.

Optimizing the Process

When optimizing a process it is critical that all variables be looked at very closely. When molding PP/TPO compounds it is very important that the mold be cooled properly, the material be properly mixed, and that the gates be of proper size and location. A careful look at these things can greatly increase your chances of meeting or exceeding your quoted cycle time.

Mold cooling

It is critical that proper mold temperatures be achieved while molding. This will require a high flow thermalator and efficient cooling lines. The use of a water manifold and flood type cooling systems are not recommended as they are restrictive and promote flow in the path of least resistance. See your selected material's processing sheet for cooling recommendations.

Material Mixing

Properly mixing the material starts with the material before reaching the feed throat and ends with the material coming out of the mold nozzles. It is important that the natural material and concentrate be blended properly with a gravimetric type blender. Once the material is in the barrel it will need a good mixing screw to mix the material. We recommend a distributive type mixing screw with a high shear spiral type mixing section on the front of the screw. Please contact a technical representative before purchasing a screw so we may talk more about the screw requirements.

Gating

It is very important to have the right size and quantity of gates when molding any plastic material. Please refer to our gating guideline before cutting any steel when it applies to gating. Our technical representatives are also a good source of information and can help with the size and location of gates. The right gate size and location can have a large effect on gate freeze and proper mold packing.

## Troubleshooting

Short Shots

When troubleshooting short shots it is important to look at your shot size, cushion, decompression , nozzle tip and injection speed and pressure. A short shot can be as simple as not having a large enough shot size to as complex as having a blown manifold. It is a problem that should be evaluated with caution. A few things to check are

1. Shot size: Make sure that your part is roughly 90-95% full before transferring and that you have an adequate cushion to fill and pack the rest of the part. You should first determine how full the part is at transfer by using no pressure for pack and hold. Then use roughly 75% of pressure at transfer to start packing and pack until gate freeze occurs.
2. Non-return valve: Ensure that your non-return valve is working properly by watching the screw during injection. If the screw rotates during injection, the valve may be letting material leak past the valve. Increasing decompression may allow the valve to seat better and allow a better shut off condition, but could cause splay or other cosmetic issues.
3. Injection Rate: If the injection rate is too slow it is very possible that the gate will freeze before the part is full. Be sure that the gate is not freezing before proper part packing by monitoring the machine and ensuring that transfer is achieved.
4. Gate design/location: If the gate(s) are in a poor location or the improper size it will sometimes not allow the part to fill properly. You will often see ribs and/or bosses that are not completely full if this is a problem.
5. Venting: An improperly vented tool will sometimes prohibit the part from being filled properly and completely.
6. Nozzle seat: Another possible reason for short shots is improper nozzle seat. Ensure that the nozzle tip orifice is slightly smaller and of the same radius (slightly smaller) as the sprue bushing on the mold. This will reduce the possibility of a “cap” on the front of the barrel.

Sinks

Sinks can be eliminated or decreased with some simple practices. It is first important to understand if it is a sink or if it is a pull.

1. Proper mold packing: Be sure to do a part weight study to determine when the gate freezes and to ensure that you are packing the most amount of material possible into the mold. This will hopefully pack out the sinks that are seen.
2. Proper gate size: It is very important to have the proper gate size and location. It can be impossible to pack out a sink if the gate is too far away or too small. Following our gating guidelines and speaking with or technical representatives can help you out with these problems.
3. Rib/Wall ratio: The rib/ boss thickness should not be more than 40% of the thickness of the nominal wall.
4. Rib/Boss texture: It is common for ribs to have a poor texture on them, which will cause them to stick in the mold. Sticking ribs in a mold will often give the appearance of a sink, but is in fact a "pull".

Splay

For the sake of argument splay will be included with silver streaking. They both appear to be the same defect, but some people differentiate between moisture splay and shear splay, calling shear splay silver streaking.

1. Proper drying: If the material you are using has talc in it, we recommend that you dry that material according the process sheet. Be cautious of over drying the material as this may cause some of the additives to be cooked out of the material. Polypropylene does not absorb moisture but the talc will, it is important to drive off that moisture.
2. Venting: Be sure to have the tool properly vented in all possible locations. Venting ejector pins and lifters is also suggested.
3. Excess shear: Too high shear can often cause silver streaking or splay. Controlling and reducing the flow rate through the area where the splay is seen can eliminate this.
4. Gas trapping: If two or more material flow fronts are converging on each other it can sometimes cause a gas trap that is eventually moved from the area and is smeared out across the part. Try to eliminate the gas trap by changing the material flow in that area.

Color Streaks

Color streaks are often a problem when the material has not been blended at the proper ratio or when the screw is not of a proper design.

1. Material blending: It is critical that the proper material/concentrate ratio be used. Any deviation from this ratio could cause problems with color streaks and/or improper color.

2. Material mixing: Proper screw design is critical when molding a PP/TPO compound. We recommend a distributive mixing screw with a high shear spiral mixing head on the tip of the screw. Slowing the screw down to where it gets to its full shot position about 2 seconds before the cure time has ended is a good practice to follow. We also ask that you consult with a technical representative before purchasing a screw to ensure you are purchasing the correct design.
3. Shear Rate: Color can sometimes separate from the material while flowing through the gate or high shear areas. Slowing the fill down or reducing the restriction can reduce the shear rate and therefore reduce color streaking.

<u>Tiger Striping</u>

The occurrence of tiger stripes comes when the material sees to high of a shear rate going through the gate and when the material has to travel a long distance.

1. Reduce Shear Rate: Opening up the gate(s) or reducing the injection rate can reduce or eliminate the appearance of tiger stripes.
2. Heat up the grained side of the tool: This will help the material flow more easily through the tool.
3. Increase Melt Temp: This will help the material flow a little easier through the gate and possibly help with the appearance of tiger stripes.

<u>Flash</u>

Flash is a common problem with injection molding and can often be remedied very easily.

1. Parting Line: Ensure that the parting line on the injection mold is properly maintained and is free of any debris.
2. Shot size: Too large of a shot size or too much pack can often flash around the gate.
3. Clamp Pressure: Insufficient clamp tonnage or pressure can result in poor shutoff, which will cause the part to flash. The available tonnage should be sufficient to mold the desired part.
4. Melt Temperature: If melt temperature is too high it can cause the material to become very watery which could cause the part to flash more easily.

<u>Bubbles</u>

Often time's bubbles can be seen in a plastic part, the following are some reasons for this occurring.

1. Venting: Insufficient venting could cause the buildup of gas, which could result in a bubble.
2. Melt Temperature: Too high of a melt temperature could cause the material to degrade slightly which would cause bubbles.
3. Mold Temperature: A high mold temperature either through the whole mold or locally could cause bubbles, this needs to be monitored closely.

 

Voids

Voids are very similar to bubbles, but less noticeable.

1. Venting: Proper venting around the parting line and on ejector pins and lifters can reduce the amount of gas that is in the mold, this will help eliminate voids.
2. Melt Temperature: A high melt temperature can cause some material degradation, which could cause some voids to appear.
3. Packing: Insufficient packing can sometimes cause voids to occur. This happens because the frozen layer of material stays against the tool while the parts sinks from the inside out.

Poor Weld Line Strength

Weld line strength is the direct relation to how the two material fronts come together and melt together to become one.

1. Melt Temperature: If the two melt fronts are not at a high enough temperature it could cause the weld line to be weak.
2. Pressure Drop: Insufficient pressure applied to the two melt fronts can sometimes cause the weld line to be poorly welded together.
3. Venting: Poor venting in the area near the weld line can cause the material fronts to encounter a restriction, which could cause a poor weld.

Brittleness

Parts being too brittle can sometimes be seen, but is quite uncommon with PP/TPO Compounds.

1. Concentrate: Be sure that the concentrate is being added at the appropriate rate. Too much concentrate can cause the parts to be very brittle.
2. Melt Temperature: A melt temperature that is very high can degrade the material to a point where it will become brittle. Be sure that your melt temperature is within the specified range for your selected material.

Gate Blush

Gate blush is very common in injection molding, more often then not, it is seen with the use of sub-gates or cashew gates.

1. Packing: An over pack of the part can cause a blush to be seen around the gate area. If the selected pressure/time is needed to fill the part out, a larger gate may be needed.
2. Flow rate: A high flow rate can sometimes result in a gate blush, slowing the injection rate down or making the gate larger can help eliminate this problem.

 

Gloss Differences

When a difference in gloss is seen, it is important to make a careful evaluation of the situation, use the following to help determine what the problem is.

1. Temperature and Pressure: If two melt fronts are coming together and the temperature is too low or the pressure is too low it will not allow the area to be packed out properly, causing a higher gloss in that area.
2. Venting: Be sure to check all vents around parting lines, on ejector pins, and on lifters. Improper venting can cause gas to come to the surface of the part and will give it a higher gloss.
3. Flow distance: If the material has traveled a long distance it will sometimes not be packed out properly far from the gate. This can be helped with a larger gate or changing the gate location to help pack that area out better.

 