OVERMOLDING GUIDE

GLS™ TPE

AVIENT™

Table of Contents

Overmolding

Overmolding is the injection molding process where one material (usually a TPE) is molded onto a second material (typically a rigid plastic). If properly selected, the overmolded TPE will form a strong bond with the plastic that is maintained in the end-use environment. The use of primers or adhesives is no longer required to achieve an optimum bond between the two materials.

Overmolding can be used to enhance many features of product designs, including:

Safety	Ergonomics	Product Functionality
• Improved grip in dry and wet environments. • Vibration damping.	• Increase in comfort level.	• Water resistant seal. • Sound absorption. • Electrical insulation.

Overmolding Process Types

Two injection molding processes dominate the manufacture of overmolded products: insert molding and multi-shot injection molding.

Insert Molding

The most widely used process is insert molding, where a pre-molded insert is placed into a mold and the TPE is shot directly over it (**Figure 1**). For molders, the advantage of insert molding is that conventional single shot IM machines can be used (new machinery expenditures are not necessary), and the tooling costs associated with insert molding are lower than with multi-shot processing.

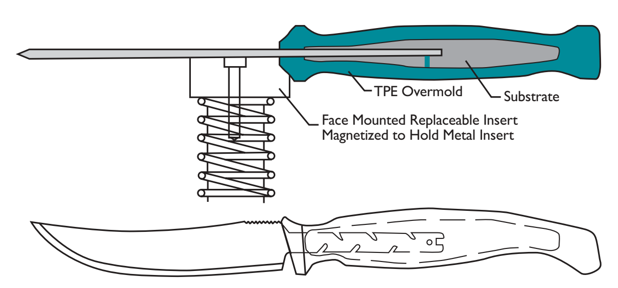

Figure 1. Spring-loaded insert for metal tolerances.

Multiple Material Molding

Multiple material, also known as two-shot (or multi-shot), molding requires a special injection molding machine that is equipped with two or more barrels, allowing two (or more) materials to be shot into the same mold during the same molding cycle.

A molder will choose multi-shot molding to reduce cycle times, achieve superior part quality and reduce labor costs.

To reduce the probability of flashing the mold, the overmold should be designed with the following guidelines:

- Provide a 0.015" – 0.030" (0.38 mm – 0.76 mm) deep groove on the substrate, along the edge of the TPE overmold (Figure 2). The steel should have positive shut-off in the groove. In addition, shrinkage of both the TPE and substrate should be considered.
- When metal or other non-compressible substrates are used, provide springs underneath the steel sections shutting off on the substrate to prevent flashing due to a steel insert with a poor fit.

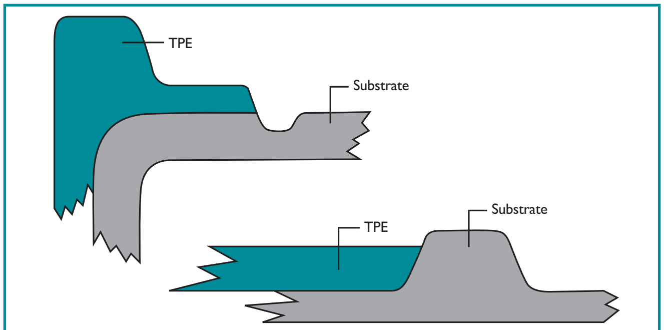

Figure 2. Examples of shut-off designs.

The fundamental bond of GLS TPE over the rigid substrate can be provided by a combination of three basic methods:

- Molecular adhesion.
- Mechanical design techniques.
- Mechanical interlocks.

Figure 3 illustrates three mechanical interlock design options that can be utilized to optimize finished component bond strength.

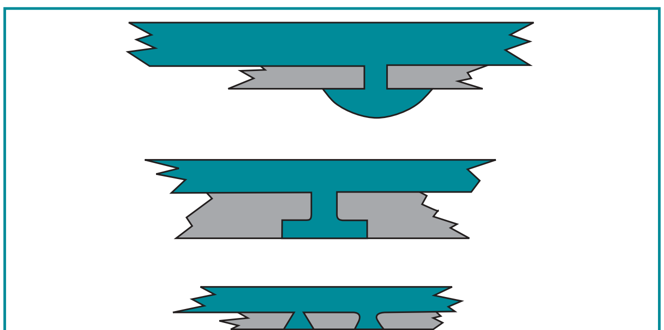
Figure 3. Types of successful mechanical interlocks.

Utilizing texture on the TPE overmold surface is a good way to impart a unique surface feel to the product and minimize the appearance of surface defects. It should be emphasized that certain textures will lead to a perceived hardness that is higher (or lower) than the actual hardness of the TPE. As a result, the TPE surface texture should be taken into consideration during the material selection phase of the product development process.

Mold Layout and Support

For multi-cavity tools, the cavity layout should be physically balanced. In a balanced system, the TPE melt flows to each cavity in equal times under uniform pressure. An unbalanced runner may result in inconsistent part weights and dimensional variability.

Figures 4 and 5 illustrate examples of balanced and unbalanced runner systems.

In insert molding applications, proper support of the plastic insert is required. Without support, the plastic substrate can deform due to the TPE injection pressure. In extreme cases, the insert will break or the TPE melt will impinge through the plastic insert. Flashing in certain areas of the tool can also result from displacement of the insert within the mold cavity. This is usually not an issue with two-shot molding because the first shot is automatically supported on the "B-half" of the tool.

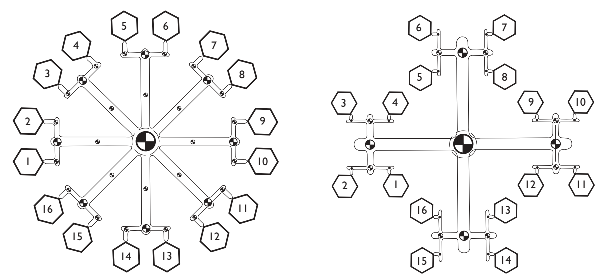

Figure 4. Balanced runner systems.

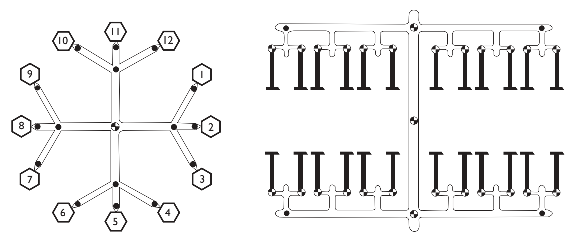

Figure 5. Unbalanced runner systems.

Venting

It has been well-established that the lack of adequate venting is a processing issue for both overmolding production and standard injection molding. If vents are not incorporated into the mold design, the adhesion of the TPE to the substrate can be critically affected in specific areas of the part where air is trapped in the cavity during injection. As the TPE melt is injected into the mold cavity, the air in the cavity must be able to effectively exit the tool. This is usually achieved via the addition of vents at the ends of flow (full peripheral venting is the best solution).

Figure 6 provides an illustration of appropriate venting.

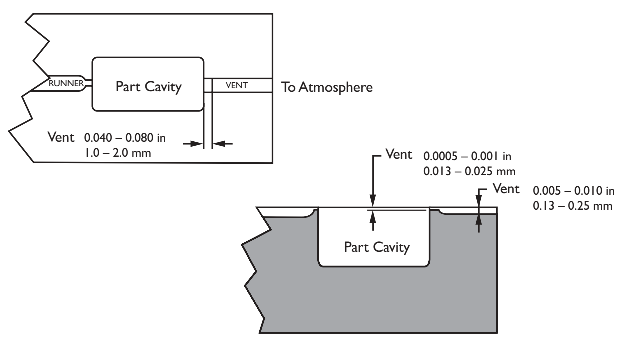

Figure 6. Example of appropriate venting.

Sprue Pullers

Sprue pullers are used to pull the sprue out of the A-half of the tool so that the plastic may be ejected automatically out of the mold using an ejector pin. For GLS materials with hardnesses over 50-90 Shore A, a sprue puller with a Z-type may be used to pull the sprue out.

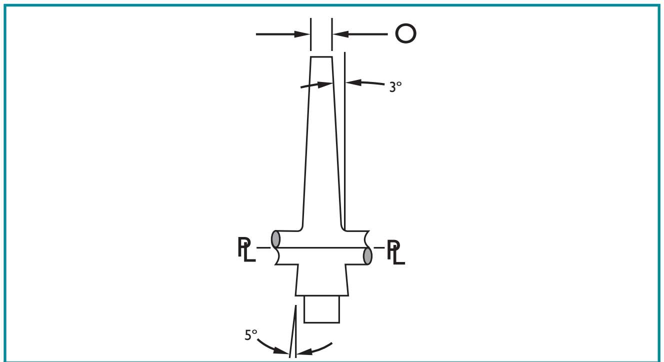

Figure 9. Conventional sprue.

The sprue puller should be polished, while the sprue and runners should have a rough EDM finish. This would enable the plastic to stick to the sprue puller as it is withdrawn. In the case of softer GLS materials, it may be necessary to use a more aggressive sprue puller such as a pine tree design (**Figure 10**). It is extremely important to position cooling channels close to the sprue puller since this can help the plastic to harden around the puller before it is withdrawn.

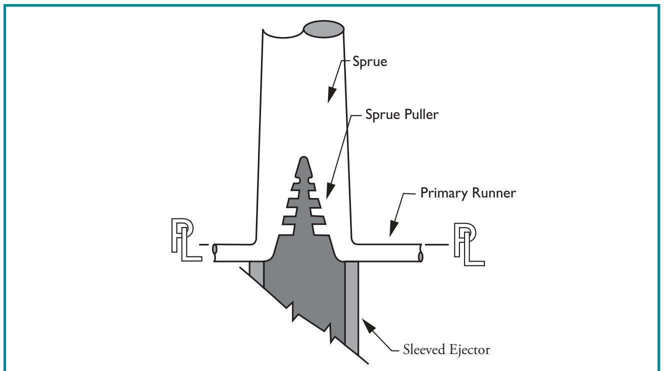

Figure 10. Pine tree sprue puller for soft GLS TPEs.

In the case of three plate tools, a sucker pin with a simple spherical end is suitable. This would help pull the runners out of the floater plate. A low reverse draft of 10-15° may be added to the sides so as to help form a slight undercut. As stated earlier, it is very important to incorporate cooling channels on either side of the sucker pin. In addition, a rough EDM or sandblast finish should be used for the runners and a polished finish for the sucker pins. This should help the sucker pin to pull the runner out more easily. A more aggressive sucker pin may pull the runners out, however it may be difficult to automatically remove the runners off the sucker pin once the mold is open. Sprue pullers may not be necessary when hot sprues are used.

Gate Design and Location

Most conventional gating types can be used in the molding of GLS TPEs.

The type of gate and the location, relative to the part, can affect the following:

- Part packing.
- Gate removal or vestige.
- Part cosmetic appearance.
- Part dimensions, including substrate warpage.

Submarine (Tunnel) Gate

Submarine or tunnel gates are self-degating; during part ejection, the molded part and runner are separated by the tool steel. To promote degating, a radius can be located at the end of the sub-gate; typical dimensions are 1.5-2 times the radius of the gate. **Figure 13** shows a typical design of a submarine gate.

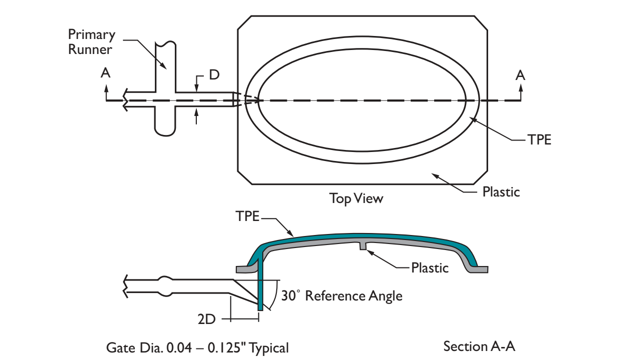

Figure 13. Submarine gate.

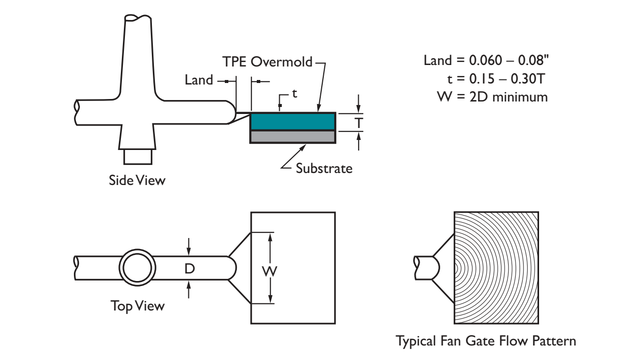

Figure 14. Conventional fan gate.

Fan Gate

A fan gate is a variation of a tab gate (**Figure 14**). The fan gate distributes material into the cavity more evenly than other gate designs. It is normally used in parts that require a high degree of flatness.

Sprue or Direct Gate

Sprue, or direct gating is often used on prototype parts; the cavity is placed directly in line with the center of the sprue. If the product design requires a sprue gate, a hot sprue is preferred over cold because it reduces scrap, decreases cycle time, and allows for easier processing.

Sprue gating is not recommended for production tools or for aesthetic parts due to the potential for "cold-slugs" on the part surface. The sprue will also need to be manually trimmed. If this type of gate is desired, both the sprue length and diameter should be as short and small as possible.

Additional Factors that Affect Adhesion

- Grade of plastic substrate (glass-filled, mineral-filled, heat-stabilized, lubricated).
- The more polymer on the substrate surface, the better the adhesion.
- Ensure that the chosen TPE is designed to bond to the substrate.
- Appropriate TPE thickness — too thin can lead to delamination.
- Use of mechanical interlocks in the component design.
- Proper shut-off design.
- Adequate venting is critical, especially at ends of flow.
- Type of color concentrate carrier used in both the plastic and the TPE.
- Pre-drying of the TPE overmold material, if required.
- Substrate preparation and cleanliness.
- Higher TPE melt temperature generally provides higher bond strength.
- Control melt temperature by injection speed, 1st stage pressure, then barrel temps (to fine-tune).

Problem: Ejector pin marks

Possible Cause	Corrective Actions
Pack pressure is too high.	• Reduce pack pressure.
Parts are soft during ejection.	• Increase cooling time. • Increase water flow rate in cooling time.
Ejection force is too high.	• Reduce mold temperature. • Texture ejector pins and mold surface for better release. • Increase size of pins. • Consider use of pneumatic poppets to assist in component release.

Problem: Flash

Possible Cause	Corrective Actions
Injection pressure is too high.	• Reduce 1st stage injection pressure and fill time.
Shot size is too high.	• Decrease shot size.
Material viscosity is too low.	• Reduce injection speed (melt velocity). • Reduce TPE melt temperature in 10˚F/C increments.
Insufficient clamp capacity.	• Increase machine clamp tonnage (min. 2 tons/sq. in.) with a larger press.
Vents are too deep.	• Reduce thickness of the vents (max. 0.001" or 0.025 mm depth).

Problem: Flow marks, folds and back fills

Possible Cause	Corrective Actions
Melt temperature is too low.	• Increase melt temperature.
Filling from thin to thick sections.	• Reposition the gate to a thick section.
Surface irregularity.	• Surface texture can be added to part design and steel wall cavities.
Uneven filling of section.	• Relocate gate to balance the flow or reduce the runner diameter.

Problem: Part sticks during ejection

Possible Cause	Corrective Actions
Pack pressure is too high.	• Reduce pack/hold pressure.
Parts are too warm.	• Increase total cycle time. • Reduce mold temperature. • Reduce TPE melt temperature.
Insufficient ejection force.	• Increase number of ejector pins and increase diameter on larger components. • Consider use of pneumatic air poppets. • Reduce 2nd stage injection pressure.
Polished ejector sleeve finish.	• Sandblast B side.

Problem: Part sticks in A-half or stationary side of tool

Possible Cause	Corrective Actions
Insufficient extraction force.	• Sandblast A side. • Run A side cooler. • Increase draft on part in A-half of tool.

Problem: Short shots, no burn marks

Possible Cause	Corrective Actions
Not enough material.	• Increase shot size, if possible. • Determine that machine barrel has enough capacity to fill TPE cavity. • Reduce RPM and back pressure.
TPE viscosity is too high.	• Increase TPE injection speed (melt velocity). • Increase TPE melt temperature.
Insufficient injection force.	• Increase 1st stage injection pressure.
Blockage at the feed-throat.	• Decrease the barrel temperature in the rear.
Vents are blocked.	• Inspect and clean vents, if required.

TPE Overmolding Troubleshooting

Problem: Flash (over substrate or on periphery of part)

Possible Cause	Corrective Actions
Poor mold fit.	• Check mold fit.
Inadequate molding machine tonnage.	• Increase machine tonnage to a minimum of 2 tons/in^2.
Improper TPE shut-off design.	• Recut tool to obtain complete shutoff with minimum 0.002" (0.05 mm) interference into substrate.
Substrate shrinkage/lack of supports.	• Check for substrate sinks and add substrate support.
Injection pressure is too high.	• Reduce 1st stage injection pressure and fill time.
Shot size is too high.	• Decrease shot size.
Material viscosity is too low.	• Reduce injection speed. • Reduce TPE melt temperature in 10°F/C increments.
Insufficient clamp capacity.	• Increase machine clamp tonnage (min. 2 tons/in^2) with a larger press.
Vents are too deep.	• Reduce thickness of the vents (max. 0.001" or 0.025 mm depth).

Problem: Poor adhesion

Possible Cause	Corrective Actions
Injection speed is too slow and melt temperature is too low.	• Increase injection speed and melt temperature. • Reselect correct grade of GLS TPE to match plastic selected.
Contamination.	• Check for color concentrate compatibility.
Incompatible materials.	• Avoid lubricated plastic grade and do not use mold release spray.

Index

1.844.4AVIENT
www.avient.com