

GLS™ TPEs

Table of Contents

	Page
1. INTRODUCTION	3
2. PART DESIGN	4
3. MOLD DESIGN	6
4. HOT RUNNER SYSTEMS.....	17
5. OVERMOLDING	20
6. MACHINE SELECTION	21
7. MATERIAL HANDLING AND PREPARATION.....	22
8. PROCESSING CONDITIONS.....	24
9. TROUBLESHOOTING.....	28

INTRODUCTION

Objective

This document provides guidelines for part design, mold design and processing of styrenic block copolymer (SBC) thermoplastic elastomers (TPEs). The Avient GLS™ product families that include styrenic TPEs are Versaflex™, Dynaflex™, OnFlex, Versalloy™ and Versollan™.

SBC Rheology

One major characteristic of SBCs is that they are shear dependent. A material is shear dependent when its viscosity is higher at low shear rates (such as extrusion) and lower at high shear rates (as in injection molding). Therefore, SBC TPEs will flow more easily into thin areas of the mold at high shear rates. The shear thinning behavior of SBCs should be considered when designing injection molds and also when setting mold conditions during processing.

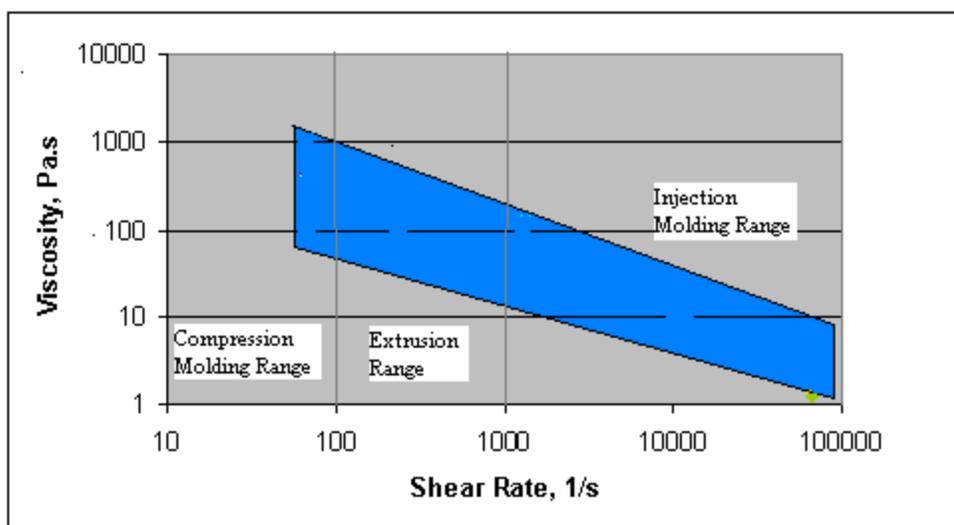


Figure 1. The effect of shear rate on the viscosity of GLS styrenic TPEs (measured at 390°F (200°C))

To obtain information regarding the viscosity of an individual grade, refer to the product technical data sheet, available at www.avient.com or contact your Avient representative.

PART DESIGN

General Part Design Concepts

When designing a TPE part, there are a few general rules to follow:

- The part wall thickness should be as uniform as possible. Transitions from thick to thin areas should be gradual to prevent flow problems, back fills, and gas traps.
- Thick sections should be cored out to minimize shrinkage and reduce part weight (and cycle time).
- Radius / fillet all sharp corners to promote flow and minimize no-fill areas.
- Deep unventable blind pockets or ribs should be avoided.
- Avoid thin walls that cannot be blown off the cores by air-assist ejection.
- Long draws with minimum draft may affect ease of ejection.

Flow Length and Wall Thickness

The maximum achievable flow length is dependent on the specific material selected, the thickness of the part, and processing conditions. Generally, GLS TPEs will flow much further in thinner walls than other types of TPEs. The flow to thickness ratio should be 200 maximum, however this is dependent on the material and the part design. High flow GLS TPEs (such as Versalloy) have been used successfully to fill flow ratios up to 400.

The measurement of spiral flow offers a comparative analysis of a material's ability to fill a part. The spiral flow test is performed by injecting a material into a spiral mold (similar to a ribbon formed into a spiral). The distance the material flows is measured in inches. In this case, the spiral flow test was conducted using two different injection speeds (3 in/sec and 5 in/sec). The typical spiral flow lengths for the various GLS product families are summarized in Table 1. With specific formulations, flow lengths of up to 40 inches (at 5 in/sec injection speed) are possible.

Table 1. Typical Spiral Flow Lengths for GLS TPEs*

Series	Flow length, in	
	3 in/sec	5 in/sec
Dynaflex™ D	13–15	18–20
Dynaflex™ G	12–22	18–30
Versaflex™	9–16	13–26

*Spiral flow tests performed using 0.0625 in thickness and 0.375 in width channel at 400°F.

For spiral flow information about a specific grade or additional details about the spiral flow test procedure, please contact your Avient representative.

Undercuts

The flexibility and elastic nature of SBC TPEs allows for the incorporation of undercuts into the part design. Because of their excellent recovery characteristics, GLS TPEs are capable of being stretched and deformed, allowing them to be pulled from deep undercuts (Figure 2). If both internal and external undercuts are present on the same part, slides or core splits may be necessary. Parts with internal undercuts (e.g. bulb-shaped parts) may be air ejected from the core by use of a poppet valve in the core. Minor permanent elongation (3%–8%) due to deformation may occur during ejection.

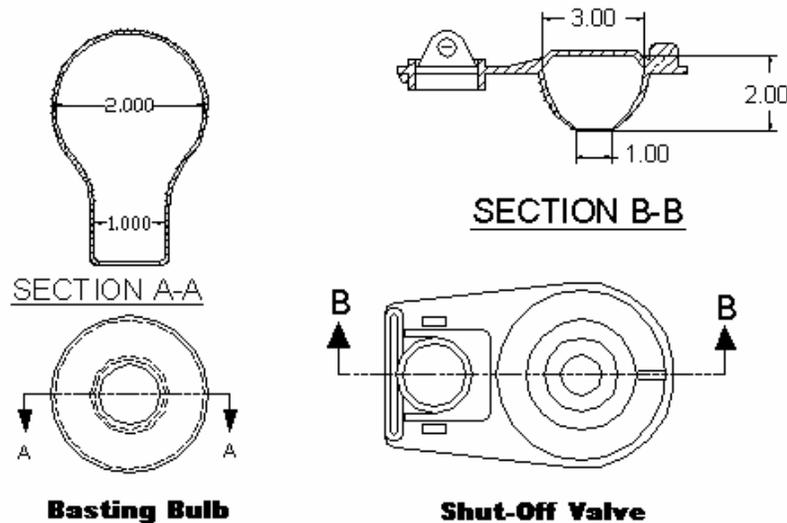


Figure 2. An example of TPE parts with large undercuts

Gate and Knit Line Locations

The product engineer should indicate the areas of the part that are cosmetic and those that are functional and include this information on the drawing. This will help the mold designer to determine the allowable gate and knit line locations.

Anisotropy

Thermoplastic materials that have different properties in the flow direction versus the cross-flow direction (90° perpendicular to the flow direction) are characterized as “anisotropic” materials. Properties that may be affected are shrinkage and tensile properties. Anisotropy is caused when the polymer chains orient in the direction of flow, which leads to higher physical properties in the flow direction. Wall thickness, injection speed, melt temperature and mold temperature are a few variables that affect anisotropy. Depending on the processing conditions and mold design, most GLS styrenic TPEs exhibit a degree of anisotropy.

Shrinkage

Due to their anisotropic nature, GLS TPEs shrink more in the flow direction than in the cross-flow direction. Generally, SEBS materials have higher shrinkage and are more anisotropic than SBS materials. Typical shrinkage values for SEBS-based formulations are 1.3%–2.5%, whereas those for SBS-based formulations are 0.3%–0.5%. Softer SEBS materials (below 30 Shore A) will shrink more than harder

materials. Some grades, such as OnFlex™ G7700s and G7800s, and Dynaflex™ G7900s contain filler, which reduces their shrinkage.

The shrinkage values reported by Avient are determined using a 0.125” thick plaque. It should be noted that shrinkage is not an exact number, but a range value. This range can be affected by the part wall thickness, melt temperature, mold temperature, injection speed, hold/pack pressures and also the time between molding and measuring. As a result, prototyping is strongly recommended for parts with close tolerances to better quantify the realistic shrinkage of a specific grade of material in a specific application.

For shrinkage values for specific grades, please refer to the product technical data sheet, available at www.avient.com or by contacting your Avient representative.

MOLD DESIGN

Types of Molds

GLS TPEs can be molded in two- and three-plate molds. Both conventional and hot runner tool designs have been used with GLS TPEs. Self-insulating hot runner tool designs are not recommended due to the potential for material degradation in the stagnation zones. Two-shot molds and insert molds can also be used. If a family mold is required, the cavity volumes should be similar, otherwise overpacking and flashing of the smaller cavity may occur.

Steel Selection

GLS TPEs are generally non-abrasive and non-corrosive. The selection of tool steel will depend on the quantity and quality of parts to be produced. For high volume production, the initial expense of quality tooling is a sound investment.

A wide variety of tool steels are available for injection mold construction. Table 2 lists the properties of common tool steels and the typical mold components for which they are used. Soft metals, such as aluminum and beryllium copper, can be used for prototype parts or short production runs up to 10,000 parts.

Table 2. Typical Tool Steel for Injection Mold Construction

Steel Type	Steel Properties	Mold Component
P-20	Pre-hardened, machines well, high carbon, general-purpose steel. Disadvantage: May rust if improperly stored.	Mold bases, ejector plates, and some cavities (if nickel or chrome plated to prevent rust).
H-13	Good general purpose tool steel. Can be polished or heat-treated. Better corrosion resistance.	Cavity plates and core plates.
S-7	Good high hardness, improved toughness, general-purpose tool steel. Machines well, shock resistant, polishes well. Disadvantage: Higher cost.	Cavity plates, core plates and laminates, as well as thin wall sections.
A-2	Good high toughness tool steel. Heat-treats and polishes well.	Ejector pins, ejector sleeves, and ejector blades.
D-2	Very hard, high wear characteristics, high vanadium content, somewhat brittle. Disadvantage: Difficult to machine.	Gate blocks, gib plates to prevent galling, gate blocks to prevent wear.
420 SS	Tough corrosion resistant material. Heat-treats and polishes well. Disadvantage: High cost.	Cavity blocks, ejector pins, sleeves, etc.

Some part designs may benefit from the use of higher thermal conductivity materials such as beryllium copper. This material is less durable than steel and may hob or wear faster than steel if used at the parting-line. Beryllium copper can be used for inserts, slides or cores to increase heat transfer rates and reduce cycle times. In cases where there is a long draw core, a fountain-type bubbler may be beneficial.

Mold Surface Treatment, Finishing and Texturing

Most GLS TPEs replicate the mold surface fairly well. To produce a glossy surface, a polished mold is required and an unfilled grade should be used. A highly polished tool and a transparent material are required to produce a part with good clarity. If a matte finish similar to that of a thermoset rubber is required, a rougher mold texture should be used (or a product such as Versalloy™ TPV alloys, which naturally produce a matte surface). In general, an EDM surface will produce a good texture and may improve release from the tool during part ejection. Matte surfaces can also help to hide any flow marks or other surface defects. Vapor honing, sand or bead blasting and chemical etching are also used to produce textured surfaces with varying degrees of gloss and appearance. To aid in release, the cavity or core may be coated with a release coating such as PTFE impregnated nickel after it has been given a sandblast or EDM finish.

Sprue and Sprue Puller Design

The sprue should have sufficient draft, from 1° to 3° to minimize drag and sprue sticking. Longer sprues may require more taper (3°–5°), as shown in Figure 3. Typically, the sprue diameter should be slightly larger than the nozzle diameter. An EDM finish is acceptable for most GLS TPE formulations.. Permanent surface lubricant treatments have also been used successfully.

Sprue puller designs vary with the hardness of the material. The different sprue designs possible and their relative dimensions are shown in Figures 4 through 7. In addition, Table 3 shows the typical hardness range for which a particular sprue design is applicable.

Table 3. Typical Sprue Designs for Various Hardness Values

Typical TPE Hardness Range	Most Common Sprue Puller Types	Figure
>50 Shore A	Tapered, Pin, Z-Type	3, 4 and 6
40-70 Shore A	Undercut	5
5-40 Shore A	Pine Tree	7

Hot sprue bushings and extended nozzles may also be used with GLS TPEs. In many molds, the sprue is the thickest wall section in the mold and will control the minimum cooling time. The use of a hot sprue, which may be viewed as an extension of the machine nozzle, can sometimes reduce cycle time. Extended machine nozzles may also be used to reduce sprue length and size. When hot sprues are used, the machine nozzle tip should be a free-flow nozzle rather than a reverse tip.

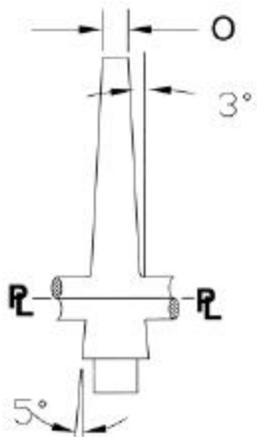


Figure 3. Tapered sprue puller

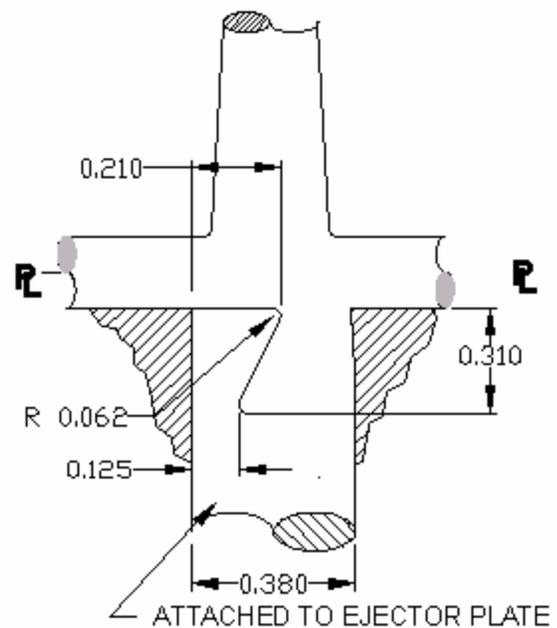


Figure 4. Z-Pin sprue puller

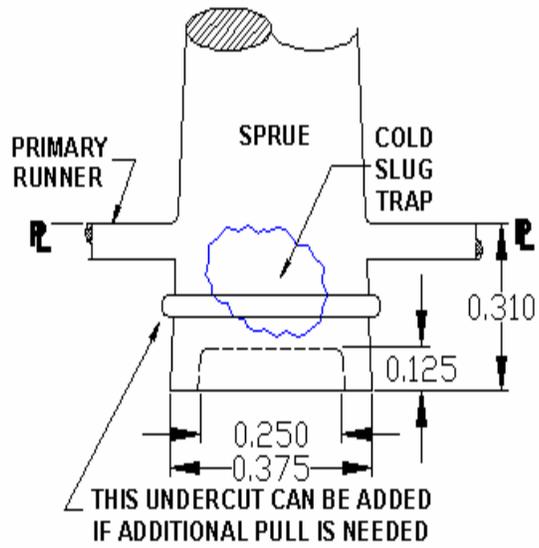


Figure 5. Undercut sprue puller

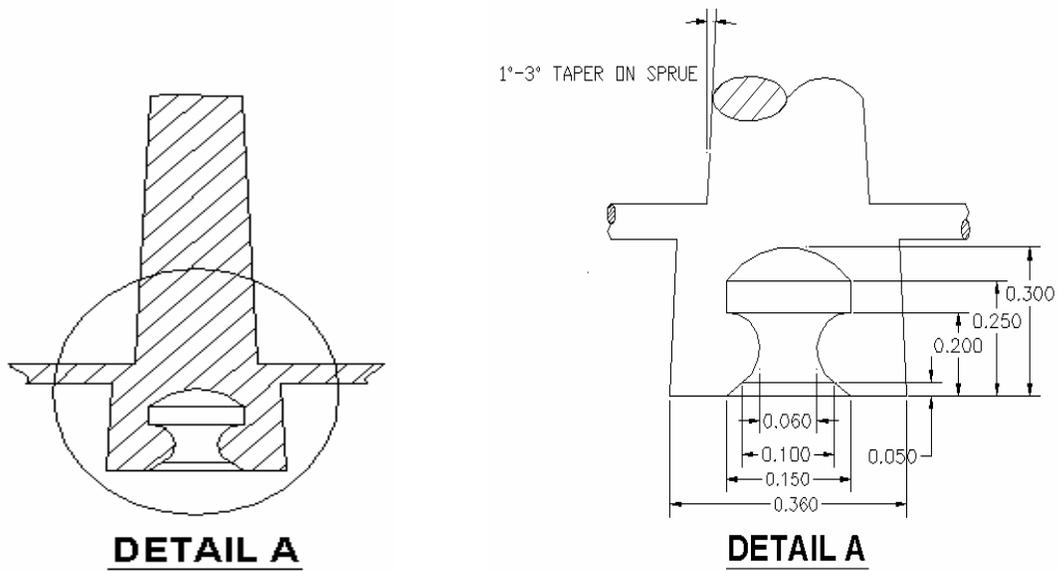


Figure 6. Sucker pin sprue puller

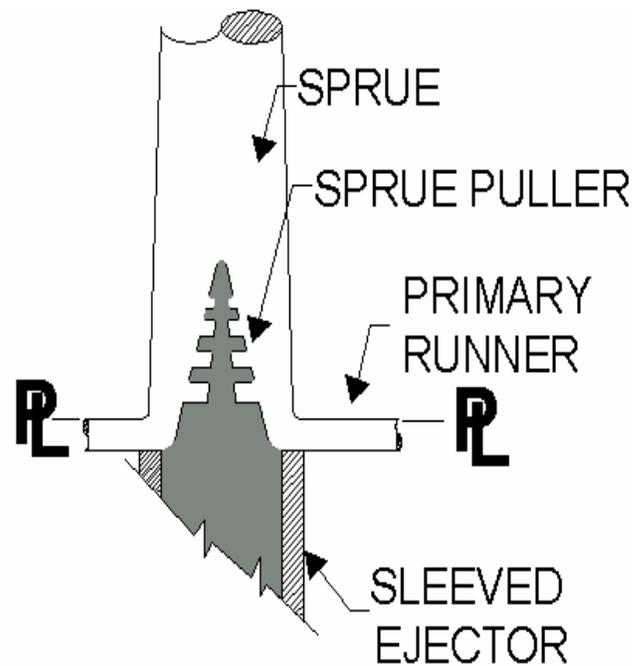
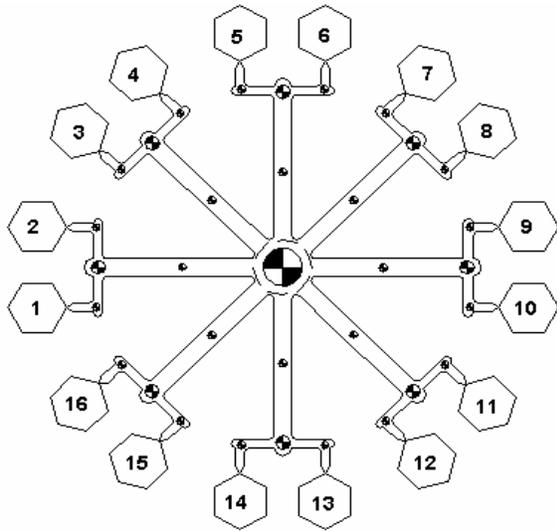


Figure 7. Pine tree sprue puller

Conventional Runner Configuration and Design

A balanced runner configuration is critical to achieve uniform part quality from cavity to cavity. In a balanced runner system, the melt flows into each cavity at equal times and pressure. The runner balance can be designed by using computer mold-flow analysis programs and verified by performing short-shot studies.

An unbalanced runner may result in inconsistent part weights and dimensional variability. The cavity closest to the sprue may be overpacked and flashing may occur. As a result of overpacking, parts may also develop high molded-in stresses, which lead to warpage. Examples of balanced runner systems are shown in Figures 8 and 9.



➤ **RUNNER "SUCKER PINS" OR EJECTOR PINS**

Figure 8. Example of balanced spider runner

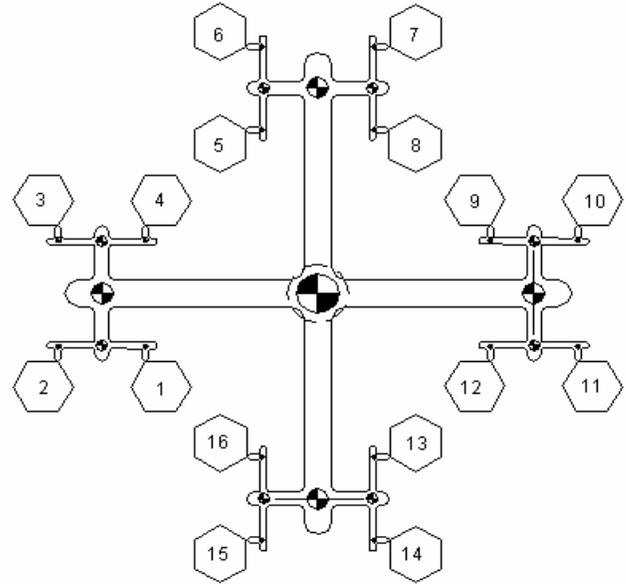


Figure 9. Example of balanced cross-runner

Figure 10 shows different runner cross-sections and their associated efficiency. Full-round runners have the least resistance to flow and surface area, allowing the material to stay molten longer. The second most efficient runner cross-section is the modified trapezoid. This runner geometry most closely simulates a full round runner but only requires machining in only one plate. Figure 11 shows typical ball cutter dimensions and the corresponding modified trapezoid runner sizes. Figure 12 illustrates typical runner dimensions.

$$W = 1.25D$$

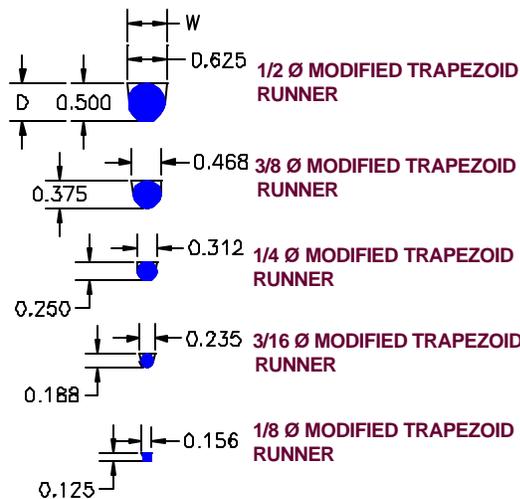


Figure 10. Typical runner cross-sections

Runner Keepers

Runner keepers or sucker pins provide undercuts to keep the runner on the desired plate but should not restrict material flow through the runner. Figures 8 and 9 show typical locations for runner keepers and sucker pins. Figure 13 illustrates an example design of a runner keeper.

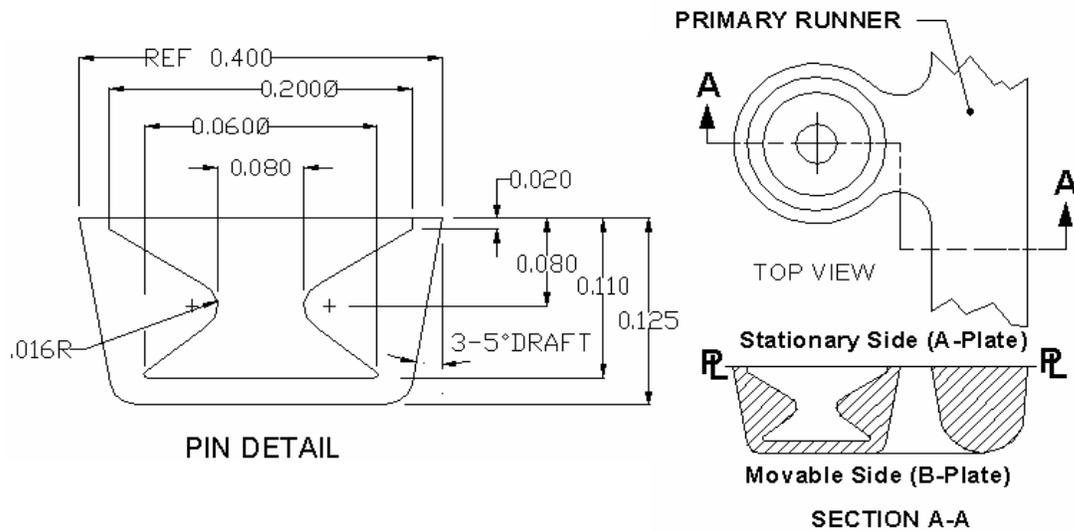


Figure 13. Runner keeper design

Gate Design and Location

Most conventional gating types are suitable for processing GLS TPEs. The type of gate and the location, relative to the part, may affect the following:

- Part packing
- Gate removal or vestige
- Part cosmetic appearance
- Part dimensions (including warpage)

The type of gate selected is dependent on both part and tool design. The gate location is equally important. To prevent the chances of jetting, locate the gate entrance in an area where the flow will impinge on a cavity wall. For automatically degating tools, the highly elastic nature of softer TPEs makes submarine gate designs or three plate tools with self-degating drops more difficult. Higher hardness and filled grades usually have lower ultimate elongation and therefore are more easily degated. To assure the gates will break at a specific location, they should have a short land length to create a high stress concentration.

Tab/Edge Gates

Tab or edge gates (Figure 14) most commonly utilize a conventional sprue and cold runner system. They are located along the tool parting line. A small undercut can be placed where the gate meets the part to minimize gate vestige caused by degating. Advantages of edge gates are ease of fabrication, modification and

maintenance. The gate depth (D) should be 15%–30% of the wall thickness at the gate entrance. Common practice is to start “steel safe”. A good starting point for the gate width should be 1.0–1.5 times the gate depth. The gate land should be equal to or slightly longer than gate depth. The gate size may also depend on the part volume. The gate area may be inserted to facilitate gate maintenance or modification.

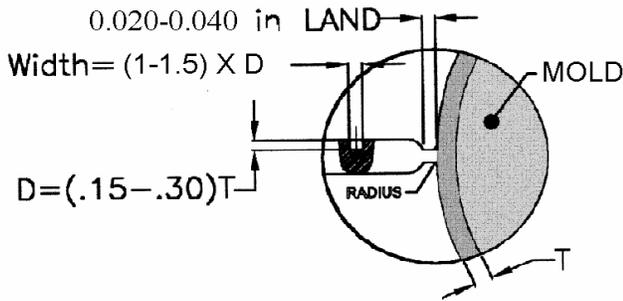


Figure 14. Tab or edge gate

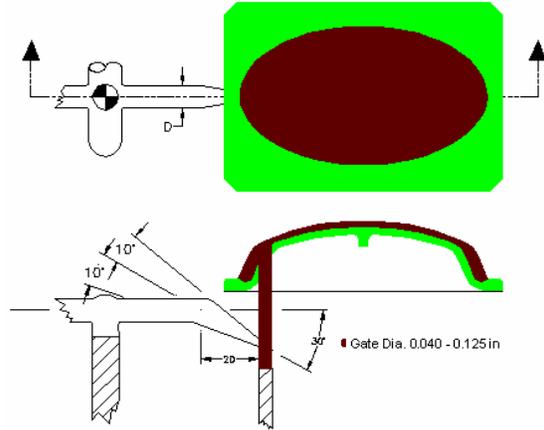


Figure 15. Submarine gate

Submarine or tunnel gates are self-degating. During part ejection, the tool steel separates the part and the runner. Figure 15 shows a typical design of a submarine gate. Cashew type submarine gates should not be used for medium to soft hardness formulations due to their high coefficient of friction and high elongation.

Fan Gates

A fan gate is a streamlined variation of a tab gate (Figure 16). The fan gate distributes material into the cavity more evenly; thus it is normally used in parts that require a high degree of flatness and absence of flow lines. It also minimizes the possibility of gate pucker or part warpage.

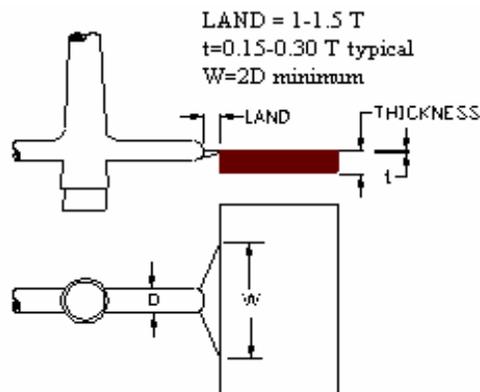


Figure 16. Fan gate

Sprue or Direct Gate

The sprue or direct gate is often used on prototype parts because it is inexpensive. This type of gating is not recommended for GLS styrenic TPEs because of their high elongation. In addition, the sprue will need to be trimmed thus appearance quality of the part is usually poor. If sprue gating is selected, care should be taken to keep both the sprue length and diameter as short and small as possible.

Diaphragm Gate

The diaphragm gate is used to maintain the concentricity of round parts. It allows even flow into the cavity and minimizes the potential for knit lines. Due to anisotropic shrinkage, flat round parts using center or diaphragm gating may not lay flat. A ring gate may also be used on the outside of a circular part.

Table 4 compares the advantages and disadvantages of the various gate types discussed in this section.

Table 4. Advantages and Disadvantages of Various Gate Types

Gate Type	Advantage	Disadvantage
Edge/Tab/Fan Gate	<ul style="list-style-type: none">• Appropriate for flat parts• Easy to modify	<ul style="list-style-type: none">• Post-mold gate/runner removal is difficult• Poor gate vestige
Submarine Gate	<ul style="list-style-type: none">• Automatic gate removal• Minimal gate vestige	<ul style="list-style-type: none">• More difficult to machine
Diaphragm Gate	<ul style="list-style-type: none">• Concentricity• Appropriate for round parts• No knit lines	<ul style="list-style-type: none">• Scrap• Post-molding gate removal
Pin gate (3-plate)	<ul style="list-style-type: none">• Automatic gate removal• Minimal gate vestige• Localized cooling	<ul style="list-style-type: none">• Requires floater plate• More scrap• Higher tool cost
Valve gate (Hot runner systems)	<ul style="list-style-type: none">• Minimal gate vestige• Positive shut-off• Minimizes post pack	<ul style="list-style-type: none">• Higher tool cost• Higher maintenance• Only for hot runner systems

Gate Location

Styrenic TPEs are anisotropic, thus they have different physical properties in the flow direction versus the cross-flow direction. Depending on the product's intended usage, these property differences could be critical to the performance of the final part. As a result, the anisotropic nature of the styrenic TPE needs to be taken into consideration when determining the gate location on the part.

The material flow may be estimated by eye or by using flow analysis programs. For higher shrinkage grades, the part may shrink near the gate, which causes “gate pucker” if there is a high molded-in stress at the gate. Parts shaped like a handle grip may warp toward the gate side of the part. Locating the gate at the top of the part minimizes this problem. Using two gates on opposite sides of the part can also address the issue, but it will result in two knit lines. If filling problems exist in thin walled parts, adding flow channels or minor changes in wall thickness can alter the flow. In some cases, it may be necessary to add a second gate to properly fill the parts.

The gate should be placed so that the flow path is as short as possible. Locating the gate at the heaviest cross section of the part can improve packing and minimize voids or sinks. If possible, the gate should be positioned so as to avoid obstructions (flowing around cores or pins) in the flow path.

The flow path of the material should minimize the possibility of formation of knit lines and flow marks. Upon injection, the material should impinge off the cavity wall to reduce the possibility of jetting. To minimize the effect of molded-in stress (at the gate) on part performance, the gate should be located in noncritical areas of the part. Also, the gate location should allow for easy manual or automatic degating.

Mold Venting

Mold venting is critical to the quality and consistency of the finished part. Venting is required to allow the air in the sprue, runner and cavity to leave the tool as the melt flows into the cavity. Inadequate venting may cause short-shots, poor surface appearance, or weak weld-lines. Potential air traps in the part design can be predicted by flow simulation software. Once the tool has been built, short-shot studies can be used to find the critical venting areas.

Vents should be placed at the last place to fill and in areas where weld lines occur. The typical vent size for GLS TPEs, is 0.0005"–0.0010" (0.012 mm–0.025 mm) with a 0.040"–0.060" (10 mm–15 mm) land. Past the land, the vent depth should be increased to 0.005"–0.010" (0.12 mm–0.25 mm) to provide a clear passage for the air to exit the tool (Figure 17). Venting in areas below the parting line can be accomplished by allowing the ejector pin to be 0,001 loose on each side (Figure 18). Venting of ribs or pockets can be achieved by venting down an ejector pin, or with the use of porous mold steels. Ejector pin vents are self-cleaning, but they should be wiped once a day to remove build-up. Porous plug vents need to be replaced, or dismantled and cleaned when they become clogged.

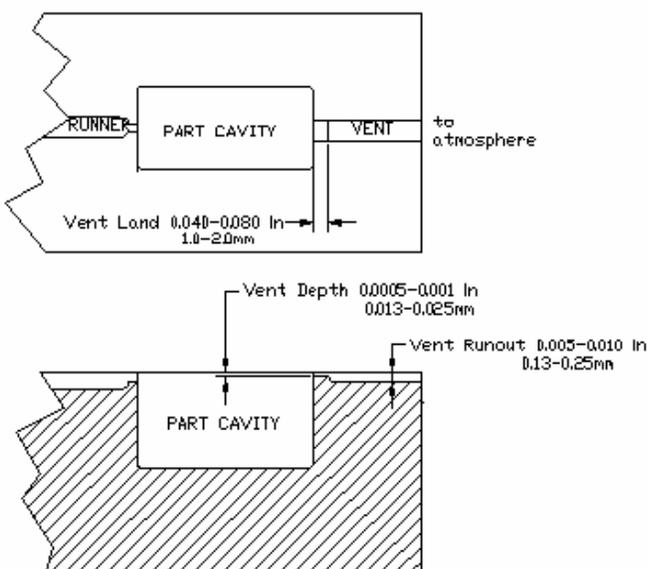


Figure 17. Mold vent design

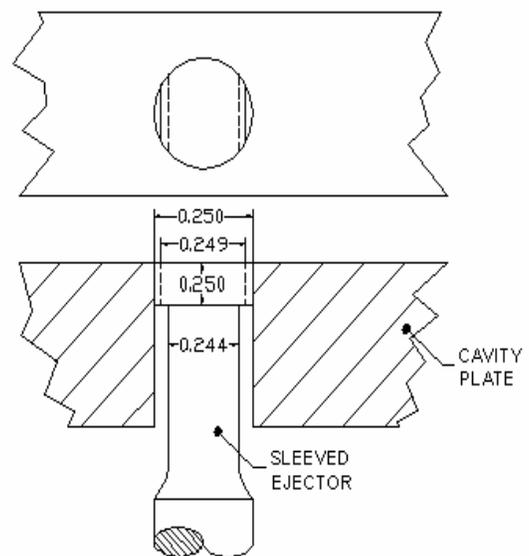


Figure 18. Venting through an ejector pin

Part Ejection

Part ejection is more difficult in long draw areas. A 3°–5° draft per side on all long draw areas is recommended. Ejector pins should be located at runner transitions and in areas of the part where appearance is not critical. The diameter of ejector pins should be as large as possible to minimize push-through marks. Larger pins also allow for easier ejection of warmer parts, which can reduce cycle time. Ejector blades, ejector sleeves and stripper rings can be used for part ejection. Air ejection and the use of poppet can help strip large undercuts, providing the material has room to deform when the air is applied. Mold surface texturing and special mold surface treatments can also help to pull the parts from the “A” half. Advancing cores are used usually when ejecting large internal undercuts.

Mold Cooling

The mold should have adequate cooling to optimize cycle time. The use of mold materials with high heat transfer, such as beryllium copper, can be used to cool slides or inserts. Commercially available fountain-type bubblers may also help to cool long cores. Separate chillers for the movable and stationary sides are suggested. This allows the processor to use differential cooling to help retain the parts on the movable (“B”) plate. Connecting cooling lines from the A to the B plate should be avoided. Special cooling for cores and slides is also an option to improve cooling efficiency.

HOT RUNNER SYSTEMS

The differences between hot runner systems; cold runners and hot sprues are summarized in Table 5. GLS TPEs are quite heat stable and are used successfully in hot runner tools today.

Selecting a particular type of hot runner system is influenced by the product design and production requirements. There are many hot runner component and tool manufacturers available. If possible, utilize a system or component supplier with experience in styrenic TPEs. **SBS TPEs can crosslink (forming gels) if they are held at high temperatures for too long a period of time, therefore hot runner tools are not recommended for these materials.**

Manifold Design

Table 5. Comparative Assessment of Hot Runner Systems

System Type	Advantage	Disadvantage
Cold runner	<ul style="list-style-type: none"> • Lower tool cost • Easily modified • Enables use of robotics 	<ul style="list-style-type: none"> • Typically governs cycle time • Potential for cold slugs • Potential for sprue sticking • Scrap (though regrindable)
Hot Sprue or Extended Nozzle	<ul style="list-style-type: none"> • Faster cycle • Minimizes scrap • Easily maintained • Better temperature control 	<ul style="list-style-type: none"> • Higher tool cost • Potential heat degradation for SBS formulations
Hot Runner	<ul style="list-style-type: none"> • No runner scrap • Faster cycle time • Precise temperature control 	<ul style="list-style-type: none"> • Highest tool cost • Purging • Material degradation • Maintenance

Externally heated systems are best. Internally heated manifolds are not suitable for TPEs. These systems typically have hot spots and stagnation zones that cause partially solidified material to cling to the cooler manifold walls. For maximum flexibility, the design should be naturally or geometrically balanced. Rheological balancing is possible, but only for a specific grade or rheometric curve. Internally heated manifolds are not suitable for TPEs – these systems typically have hot spots and stagnation zones that cause partially solidified material to cling to the cooler manifold walls. All passages should be highly polished circular cross sections with gentle bends to minimize the possibility of stagnation zones. In order to maintain high shear, minimize residence times and promote flow, the passages should have a diameter of 0.250” to 0.375”. Individualized zone controls for the hot runners are recommended and allow the operator to adjust the balance slightly to make the parts more uniform.

Hot Runner System Gates

Valve Gates

Valve gates offer the best solution for high production parts where surface quality is critical, such as medical and cosmetic products. Since valve gates leave only a slight ring on the part, gate vestige is minimized. Further improvement can be obtained by positioning the valve recessed below the part surface or concealing the gate in the part detail for aesthetic products. An example of a hot runner system with a valve gate is shown in Figure 19.

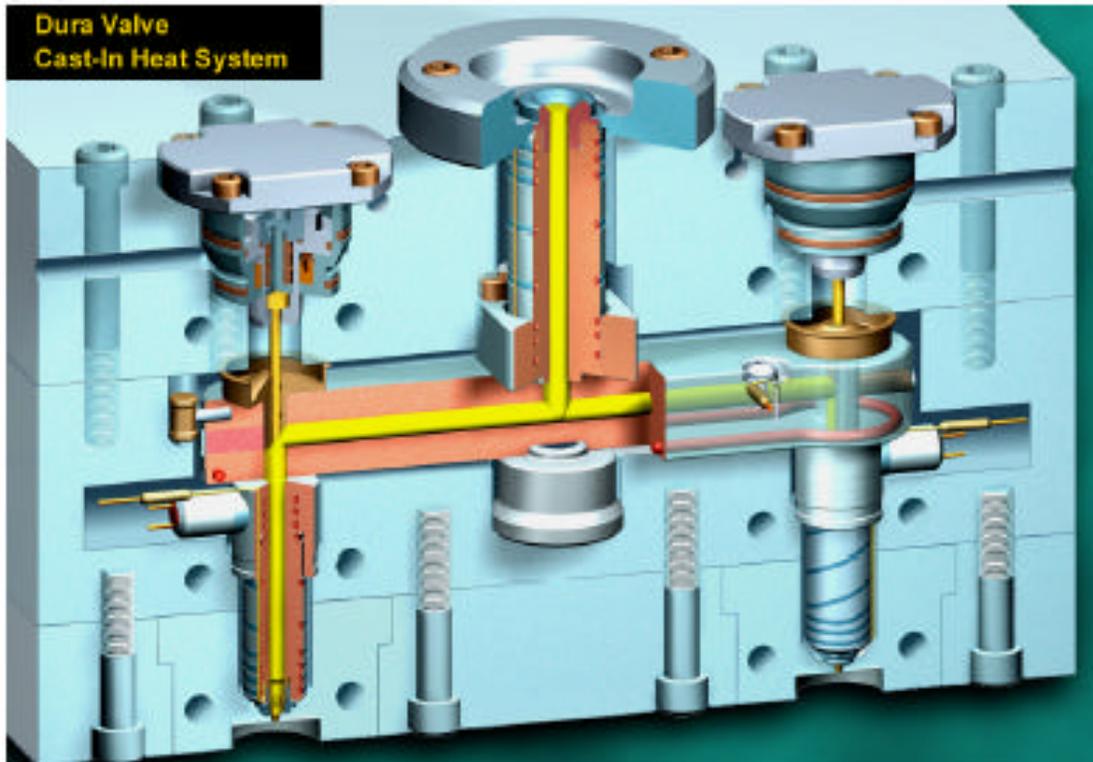


Image provided by Mold-Masters Limited, Dura is a registered trade-mark of Mold-Masters Limited.

Figure 19. Hot runner system using valve gate

The gate diameter of a valve gate should be approximately 0.030" to 0.125", depending on the size and thickness of the part. Valve gates do not require the material in the part to freeze before the valve is closed and hold pressure is released. Therefore, the screw recovery for the next cycle can start earlier and the total cycle time may be decreased. For very thick wall parts with the potential for sinks or shrink voids, valve gates can be held open for an extended time to supply make-up material and eliminate voids and sinks.

Valve gate elements need to be insulated from the mold plates to maintain proper temperature control. Only valve gates can be used for multi-cavity foam molding or cascade molding to fill long thin flows without knit lines. Due to the low viscosity of some GLS TPEs, properly maintained tight valve gates are required to prevent leakage or hair flash. Valve gates may be pneumatically or hydraulically activated. Individual heater controls at each gate will allow fine control of the melt viscosity and filling.

Hot Tip Gates

Hot tip gates are suitable for GLS TPEs but will leave some gate vestige (which can be as high as 50% to 75% of the gate diameter). Vestige can be minimized by slightly recessing the gate below the part surface. The land length of the hot tip should be less than the diameter of the gate.

The elements of the hot tip should be properly insulated from the mold plates and cavity. In order to achieve this, the land length of the gate may need to be lengthened and a portion of the land should be part of the cavity. All passages within

the tip should be highly polished and streamlined to minimize stagnation and degradation zones. The efficiency of the design may be verified by recording the time it takes to make a complete color change while producing parts. This demonstrates whether there is any residual dead zone material that continues to enter the melt stream.

For hot tip gate systems, there should be a delay long enough for the part to set up completely before mastication is initiated for the next cycle. Without a delay, the parts may become overpacked. This is particularly important for low hardness, high flow materials. To reduce overpacking for thick-walled parts with large gates, use minimal back pressure during mastication.

Since TPEs are slightly compressible in the molten state, larger runner volumes can cause hot tip gates to drool after the mold is opened. To prevent drool, the runner system should be minimized and the melt decompressed before the mold opens.

Hot tips may be used to fill a secondary cold runner to supply material to multiple parts. Each hot tip gate should use an individual temperature controller. If the hot tip manufacturer selected does not have prior SBC material experience, it may be necessary to experimentally determine the best gate type and geometry through prototyping.

OVERMOLDING

One of the largest areas of growth for TPEs is overmolding. Many product designers are utilizing TPEs to add a “soft touch” to a rigid material. GLS formulations can be overmolded onto many different substrates to alter the surface feel, improve aesthetics, and cushion against vibration – the possibilities are limitless.

Most Versaflex™, Dynaflex™ and OnFlex™ materials (and Versalloy™ TPV alloys) are suitable for two-shot or insert molding onto a PP (and in some cases, PE) substrate. Specific Versaflex™ OM grades have been specially formulated to bond to PC, ABS, nylon 6/6, PC/ABS, and PPO. The Versollan™ OM series are TPU alloys specifically designed for thin-wall overmolding (both insert and two-shot molding) onto PC, ABS, and PC/ABS substrates.

With new innovative technologies, Avient continues to develop TPEs that bond to a variety of substrates. For additional information about the standard Versaflex OM series and the development of new TPEs that bond to unusual substrates, please contact your Avient representative.

For more information about overmolding part design, mold design and processing, please refer to the “GLS™ TPEs Overmolding Guide”, available at www.avient.com or by contacting your Avient representative.

MACHINE SELECTION

Machine Types

Reciprocating screw machines are recommended. However, ram or plunger equipment has been used to produce SBS parts. Newer machines with a computer interface offer improved process control and are preferred for multi-cavity tools and high production applications. Machines with the capability of programmable injection rates and pressures can produce better quality parts. Molding machines that control the shot size by position are preferable to machines that can only control by pressure and/or time. A vertical press with a rotary or shuttle table works well for insert molding. Multi-station rotary presses allow more cooling time for molding foamed parts.

Clamp Capacity

The clamp capacity necessary for GLS styrenic TPEs is lower than most TPEs. Clamp capacity can be calculated by the following equation:

$$C = (1.5 \text{ to } 3) \times A_t$$

A clamp capacity example calculation is shown in Figure 20.

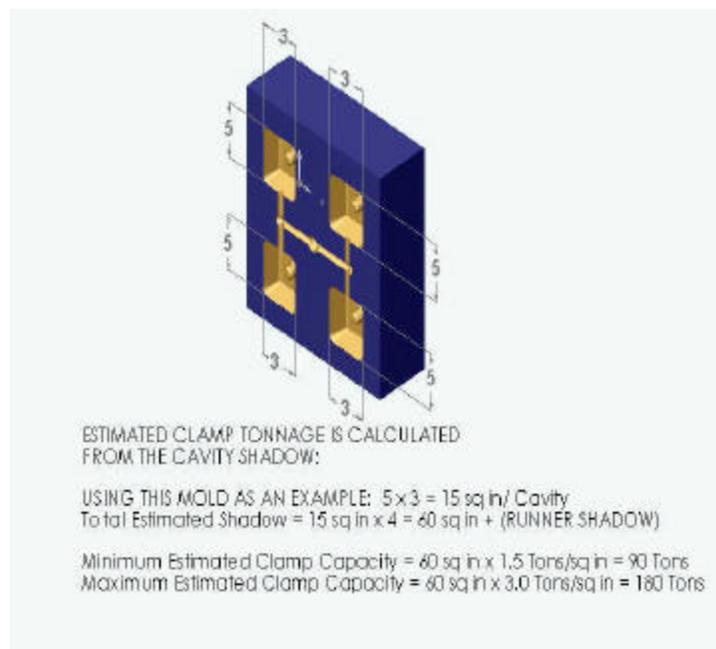


Figure 20. Estimating machine clamp capacity

Barrel Capacity

If possible, use a machine that utilizes 25% to 75% of the barrel shot size capacity. This allows for optimum temperature control of the material and minimizes material residence time at high temperatures. The material residence time for SEBS formulations should be no more than 10 minutes. The material residence time for SBS formulations should be no more than 8 minutes maximum. If a high level of regrind is utilized, the barrel capacity should be reduced.

Nozzle Selection

Smaller nozzle diameters are recommended because they promote shear heating during injection and generate less cold slug material. Suggested starting diameters are 0.0625"–0.1875" (1.59 mm–4.76 mm). Static mixing nozzles have been used to improve color dispersion for concentrates with a high letdown ratio. Extended nozzles have also been used to decrease the length of the sprue (resulting in less scrap). If a foaming agent is required (to produce foamed parts), a mechanical shut-off nozzle must be used to control foaming activity and to prevent drooling.

Screw Selection

General-purpose screws are suitable for GLS TPEs. Compression ratios of 2:1 to 3:1 are commonly used for both SEBS and SBS formulations.

MATERIAL HANDLING AND PREPARATION

Drying

Drying is not typically required for GLS TPEs. Certain specialty products, such as some Versaflex™, Versalloy™ and Versollan™ grades are hygroscopic; thus they need to be dried prior to molding. A desiccant dryer with a -40°F dewpoint is strongly recommended for drying hygroscopic materials. Specific drying temperatures and times can be found on the technical data sheets for each individual product.

Coloring

SBC TPEs have inherently superior color than most other TPEs. Therefore, they require less color concentrate to achieve a particular color and the colors produced are cleaner (less yellow) than other TPEs. Generally, the color concentrate should be lower in viscosity (have higher melt index) than the base compound. This will promote ease of dispersion.

- Styrenic (PS) color carriers are recommended for the SBS formulations.
- Polypropylene (PP) carriers are recommended for the harder SEBS formulations.
- For soft SEBS formulations low-density polyethylene (LDPE) or ethylene vinyl acetate copolymer (EVA) have been used. PP carrier is not recommended for softer grades, as the compound hardness will be affected.

Liquid colors can be used but the carrier should be a paraffin type mineral. Polyvinyl chloride (PVC) plasticizers, such as dioctyl phthalate (DOP), should not be used as carriers. Dry colors have also been used but may require more material and time to perform color changes.

The use of a polyethylene (PE) carrier may adversely effect adhesion to the substrate for some overmolding applications. If using a specialty overmolding grade, follow the coloring recommendations given on the individual product technical data sheets.

Gloss and Clarity

GLS TPEs are available in opaque, translucent and water-clear grades. The Versaflex™ CL series is formulated for high clarity. Clear formulations can produce the best metallic or pearlescent colors. High gloss clear grades have higher COFs and more intimate mold contact and therefore are more difficult to eject. Filled opaque formulations are more difficult to color in deep intense colors, but will produce good pastel colors.

Regrind

Up to 20% regrind may be used for SEBS TPEs. High levels of regrind are better tolerated in black materials. Natural, light-colored or clear formulations will more easily show contamination or discoloration. Organic pigments used to produce yellow, red, blue and green colors are more likely to change color after prolonged residence time or high regrind levels. For SBS formulations, the regrind should be kept below 25%.

GLS TPEs may have high elongation and good tear strength and therefore require the use of a high quality grinder with sharp knives. For lower durometer styrenic formulations the clearances should be set to 0.003” maximum. Only grinders with high quality support bearings and a rigid frame can maintain the tolerances necessary to achieve the necessary rotor knife to bed knife clearances. The use of a small amount of a dusting agent such as talc or calcium carbonate can minimize agglomeration during the grinding process. Feed small amounts of parts into the grinder at one time to minimize heat buildup, which can lead to agglomeration.

To allow the best incorporation of the regrind into the virgin material, the screen size should be chosen to yield particles that are roughly the same size as virgin pellets.

Purging

If the press is down for more than 10 minutes, purge before restarting production. To prevent flashing, restart the machine using a reduced shot size and gradually increase it back to the original shot size. This will help to prevent flashing from occurring behind slides or inserts. For SBS formulations - if a machine is to be left at temperature longer than one hour, purge with LDPE or polystyrene before shut down. For SEBS formulations - if the machine is down over the weekend, purge with a high molecular weight (low or fractional melt flow) LDPE at low temperatures before shut down. On start up, retract the extruder and air purge it well before attempting to fill the mold.

PROCESSING CONDITIONS

Introduction

This section describes general processing guidelines for GLS TPEs. The specific starting conditions for each individual product are located in the product technical data sheet.

Setting Barrel Temperatures

Figure 21 shows typical starting barrel temperatures. Barrel temperatures should be set progressively. The feed zone temperatures should be set fairly low [typically 250°F–300°F (120°C–150°C)] to avoid feed-throat bridging and allow entrapped air to escape. Lower temperatures in the transition zone allow proper compression and shearing of the TPE before it fully melts. To improve mixing when using color concentrates, set the transition zone temperatures above the melt temperature of the concentrate. The zone nearest to the nozzle should be set close to the desired melt temperature.

After the process has stabilized, the actual barrel temperatures should be compared to the set points. If the actual temperature exceeds the set temperature, then shear heating has caused the material to overheat. If good parts are being produced, the temperature settings should be reset to be the same as the actual temperatures.

The heaters should demand power 25% to 50% of the time. If the heaters are on continuously, there is not enough heat being produced from shear. To increase shear heating, increase the screw rpm and back pressure.

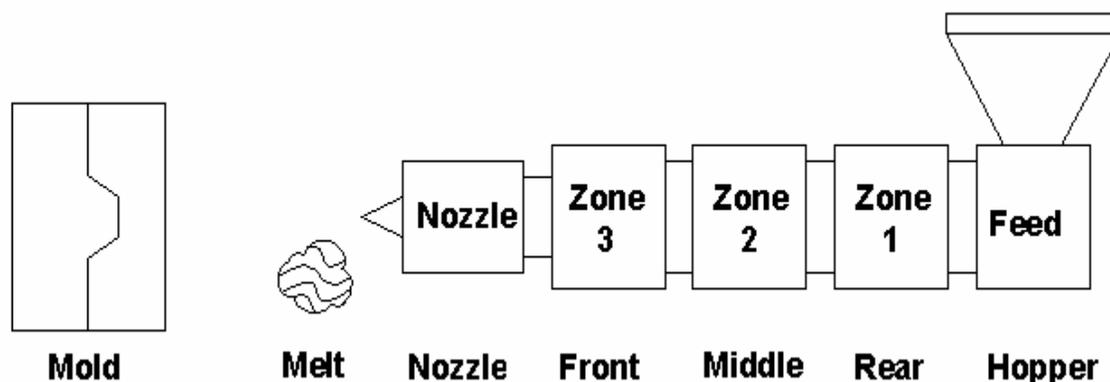


Figure 21. Suggested initial start-up conditions for injection molding

Setting Mold Temperatures

Mold temperatures should be set above the dew point temperature in the molding area. This prevents sweating of the mold and possible water contamination in the cavity. Water contamination usually appears as streaks in the parts. Mold temperatures may be raised if there are long or thin sections of the part that have proven to be difficult to fill. Higher mold temperatures usually result in higher cycle times but may improve weld line integrity and part appearance.

Table 6. Product Family and Temperatures

Product Family	Mold	Melt	Nozzle	Zone 3	Zone 2	Zone 1	Feed
SBS Formulations	75-90°F (25-32°C)	370-390°F (190-200°C)	370-390°F (190-200°C)	360-380°F (185-195°C)	340-360°F (170-182°C)	300-330°F (150-165°C)	100-150°F (40-65°C)
SEBS Formulations	110-130°F (43-55°C)	370-430°F (190-220°C)	390-430°F (200-220°C)	390-430°F (200-220°C)	370-390°F (190-200°C)	350-370°F (175-190°C)	100-170°F (40-75°C)
Supersoft Formulations	110-130°F (43-55°C)	340-390°F (170-200°C)	360-390°F (180-200°C)	360-390°F (180-200°C)	335-375°F (170-190°C)	300-330°F (150-165°C)	100-120°F (40-50°C)

Setting Shot Size

When starting up a new mold, begin with short-shots, then gradually increase the shot size until all part cavities are 80-90% filled. This procedure can minimize the potential for overpacking and prevent flash in vents. The screw position should be noted and used to set the transfer point. Monitor the cushion to insure that it is maintained during the pack and hold phase. If there is no cushion, the pack pressure cannot be maintained and there is no control of part densification. After the gate freezes, any additional material volume or pressure will just pack the sprue and runner system, which can cause difficulties with sprue removal during part ejection.

Screw rpm, Back Pressure and Screw Delay Time

The screw rpm should be set so that the screw is fully recovered for the next shot, typically 2 to 3 seconds before the mold opens. Typical screw speeds range from 50 rpm to 150 rpm. If the screw recovers too fast, and the machine is equipped with a screw delay timer, set the delay time so that there is minimal delay after the screw is fully recovered and the mold opens. This will reduce material residence time at temperature and dead time in the barrel.

Increasing the back pressure increases shear heating of the material. Normal settings for back pressure are 50–150 psi. When mixing color concentrates, higher back pressure is preferred to achieve optimum dispersion.

Injection Speed

If possible, profile the injection speed to fill the runner system rapidly and then slow down after the material starts flowing through the gate and into the cavity. Maintain this speed until the part is 90% full and then reduce it further to completely fill the cavity without flashing the part. As stated earlier, GLS TPEs are shear responsive. If a part has difficulty filling, increase the injection speeds before increasing temperature. The injection time to fill the part should be between one and two seconds. Slower fill rates may be required if surface flow defects occur.

Injection and Transfer pressures

If the machine is not capable of being controlled by fill speed, set the injection pressure high enough to fill the runner system and cavity in about 1 to 5 seconds. Adjust the initial transfer pressure to approximately 50% of the injection pressure required to fill the part cavity. This helps to minimize the pressure during the pack and hold phase of injection. When setting the shot size, monitor the cushion to insure it is maintained during the pack and hold phases.

Transfer From Boost to Pack to Hold

Newer molding equipment provides additional options for transferring from injection boost (first-stage Injection) to the pack and hold phase. The most accurate method to transfer from boost to pack pressure is by screw position. Using screw position allows the processor to consistently inject a specific volume of material to the cavity. It also provides accurate control of part packing and densification, which can help prevent sinks and voids in the part. Time is another method for controlling transfer but is not recommended. Transfer using cavity pressure is expensive because it involves installing pressure transducers in the part cavity. This process is used when highly accurate molding tolerances are required. Reducing the transfer pressure from boost to pack and hold will help to control drool at the bushing tip. If the injection unit is equipped with a profiled pack and hold phase, it can be used to reduce the velocity and pressure to the runner.

Injection Time

The optimum time to fill the runner system is approximately 0.5–1.5 seconds. It should take another 1–5 seconds to fill the cavities. If possible, it is better to control the fill time by controlling the injection speed.

Hold Time

The hold time should be set to achieve gate freeze. Usually, the gate size is the determining factor for hold time. The larger the gate the longer the hold time required to achieve gate freeze.

Cooling Time

The cooling time is principally dependent on the temperature of the melt, the wall thickness of the part and cooling efficiency. In addition, the material hardness is a factor. Harder grades (>50 Shore A) will set up faster in the mold compared to very soft grades (<20 Shore A). For an average part and medium hardness SEBS formulation, the cooling time will be approximately 15 to 20 seconds for every 0.100" of wall thickness, provided there is cooling available from both sides. Overmolded parts will take longer to cool because they can be effectively cooled over a smaller surface area. The cooling time for overmolded parts will be approximately 35 to 40 seconds for every 0.100 inches of wall thickness.

Maintaining Cushion

Cushion should be maintained or there will be no control of part densification or compensation for material shrinkage. Inadequate cushion and hold pressures will result in underfilled parts with voids or sinks and poor physical properties. Gates that freeze off too quickly (as a result of being too small or too cool mold temperatures) may also cause these above issues.

A worn or contaminated check ring can limit the machine's ability to hold pressure and maintain a cushion. GLS TPEs have lower viscosity (higher flow) than traditional thermoplastics and will leak back easier than other materials. The sealing capability of the check ring should be verified by observing the machine's ability to maintain a cushion position.

Process Conditions for Hot Sprue Bushings or Extended Nozzles

Hot sprue bushings should be viewed as an extension of the machine's nozzle. Externally heated sprue bushings work well with GLS TPEs. Hot sprues and extended nozzles are normally set 5°–10° F higher than the front zone temperature. Avoid the use of Variac or variable resistance controllers, as these controllers tend to overheat the material during short shutdowns.

Effects of Molding Conditions

If a part is molded at too low a temperature, it will require excessive pressure to fill the cavity. This will result in high molded-in stresses. This in turn can cause the part to warp during ejection or at a later time when it is exposed to an elevated temperature. There also may be greater than normal post mold shrinkage and a reduction in the ultimate elongation.

The effects of overpacking the part may include:

- Gate bulge
- Increased density, thus higher part weights
- Increased hardness

The effects of underpacking a part may include:

- Gate pucker
- Voids and/or surface sinks
- Reduced physical properties
- Lower than normal hardness

Monitoring part weights has successfully been used to verify process stability and consistency. It should be noted that the gate size/location, runner dimensions and other aspects of the mold design might also affect the properties of a part.

For clear formulations, parts that have been processed at too low a temperature will have a frosty surface appearance. SBS-based TPEs will develop a yellow or orange color and distinctive odor when they have been processed at too high a temperature or held at temperature for too long. The color and odor are strong signs that the material has been degraded. Degradation results in poor appearance and reduction in physical properties. SEBS TPEs that have been processed at too high a temperature will have a burnt odor (degradation) and in the worst case become tacky and bleed oil.

Molding Soft TPEs (1 20 Shore A)

Soft GLS TPEs have very low viscosity (high flow); thus they require minimal injection pressure. Typical values for injection pressure are 150 psi–450 psi. Most of the GLS soft TPEs are either water-clear or translucent. Clarity in the molded part can be marginally improved by increasing the mold and melt temperatures. A high polish

mold surface finish is usually justified with these products, as they duplicate the mold surface quite well.

The softer materials exhibit some tacky behavior. Cleanliness in close proximity to the molding area is important as softer materials attract and retain dust and contaminants. This tackiness also makes part ejection more difficult. In these cases, robotic sprue pickers, runner keepers, or air ejection may be required. The addition of a slight surface texture to the mold can help to mask possible surface blemishes in the molded article.

Molding Hard TPEs (³ 50 Shore A)

The harder GLS TPEs usually have higher viscosity and may require slightly higher injection pressure (400 psi–800 psi) to fill the cavity. Due to their higher modulus, hard formulations require less aggressive sprue-pullers. They also set up faster and are more easily ejected, which may result in reduced cycle times compared to softer materials.

TROUBLESHOOTING

Approach to Troubleshooting

Proper troubleshooting should use a systematic approach to resolve problems. There are two kinds of issues: those involving “quality control” and those encountered during startup.

“Quality control” issues occur when parts have been successfully produced in the past, but are now out of specification. These problems are the result of something in the process changing. To resolve them, you must determine what has changed and restore the process to its proper balance. Startup problems occur during the launch of a new tool or machine.

Quality Control Issues

Most quality control issues are caused by process conditions, the material, or machine and tool maintenance. If the problem occurred after a change in the material lot number, try a different lot of material. It is advisable to retain a store of the previous lot of material.

If the mold setting is from a previously successful part, check the original setup conditions. If the mold is used in a different machine, minor adjustments to account for the new machine may be necessary. If the mold was modified during its downtime, process adjustments may be required.

Startup Issues

To resolve startup problems, the material’s process window must be determined to ensure that there is a set of conditions that can make good parts. Start by setting the process conditions to the middle of the material’s process range and then adjust the process to fix any observed problems. If successful parts are not possible, determine what combination of variables must be changed to resolve the problems. These changes may include material selection, machine selection, and/or tool redesign.

Injection Molding Troubleshooting

QUALITY CONTROL ISSUES

OBSERVED PROBLEM	POSSIBLE CAUSE	CORRECTIVE ACTION
<i>Shorts</i>	Inadequate shot size. Vents blocked.	Material blockage at throat - lower feed zone temperature. Clean vents.
<i>Flashing</i>	Too much material. Clamp pressure low. Injection pressure too high. Injection speed too fast.	Check shot size. Check for dirt on the mold parting line of low hydraulic pressure. Check injection pressure and speed.
<i>Burns</i>	Dieseling in the mold.	Clean vents.
<i>Non-Uniform Color</i>	Poor concentrate mixing or dispersion.	Check barrel temperatures. Check back pressure and screw rpm, increase if necessary.
<i>Off-Color Part or Odor</i>	Contamination. Material degradation.	Check for contamination. Reduce the level of regrind. Reduce the rear zone temperature. Reduce the residence time. Add delay to screw recovery.
<i>Dimensions Out of Specification</i>	Loss of control of shrinkage.	Check hold pressures. Check melt and mold temperatures. Check that cushion was maintained.
<i>Surface Defects (Streaks)</i>	Moisture in part. Possible mold sweating.	Dry material. Check for contamination. Raise mold temperature.
<i>Voids or Sinks</i>	Inadequate amount of material.	Make sure there is a cushion maintained. Increase shot size. Increase pack and hold pressures. Increase hold time.

Injection Molding Troubleshooting

STARTUP ISSUES

OBSERVATION	CAUSE	SOLUTION
POOR KNIT LINES	Poor venting.	Place vents at last place to fill and at converging flow fronts Increase size of the vents.
	Too cool a melt.	Increase the melt and/or mold temperature. Increase injection speed.
PART SHORTS and has burn marks	Gas trapping / poor venting.	Improve venting and/or relocate to burned area.
PART SHORTS No burn marks	Not enough material.	Increase shot size, if possible. If not, move to a larger machine.
	Blockage in flow at the feed-throat.	Reduce rpm and back pressure. Decrease the barrel temperature in the rear.
	Not enough injection pressure.	Increase injection pressure.
	Material too high in viscosity.	Increase injection rate. Increase process temperature. Use a lower viscosity TPE. Increase gate and runner size.
PART FLASHES	Too much injection pressure.	Reduce injection pressure and time.
	Too much material.	Decrease shot size.
	Material viscosity too low.	Reduce injection speed. Reduce process temperature.
	Tool too loose.	Machine or dress the parting line.
	Not enough clamp capacity.	Use a higher viscosity TPE. Increase clamp tonnage. Reduce thickness of the vents. Move to a larger machine.

Injection Molding Troubleshooting

STARTUP ISSUES (CONT.)

OBSERVATION	CAUSE	SOLUTION
<p>VOIDS OR SINKS Voids occur inside the part. Sinks pull away from the mold wall.</p>	<p>Material shrinkage & insufficient supply of molten material.</p> <p>Not enough pack pressure during material solidification.</p>	<p>Decrease wall thickness.</p> <p>Increase the pack pressure and time. Increase the gate size. Relocate the gate to the thickest section. Increase the runner size.</p>
<p>BUBBLES Part surface bulges above a bubble.</p>	<p>Gas entrapment.</p>	<p>Relocate the gate or modify the flow path. Add a pin in the area to eliminate the gas trap.</p>
<p>FLOW MARKS Back fills Shadowing Folds</p>	<p>Filling from thin to thick sections.</p> <p>Surface irregularity.</p> <p>Uneven filling of section.</p>	<p>Reposition the gate to a thick section.</p> <p>Radius dimples.</p> <p>Relocate gate to balance the flow or reduce the runner diameter.</p>
<p>JETTING</p>	<p>High viscosity flow.</p>	<p>Increase process temperature. Increase injection speed. Decrease gate size. Change type of gate. Relocate gate to impinge.</p>
<p>WARPED PARTS</p>	<p>Anisotropic shrinkage.</p> <p>High molded-in stress.</p>	<p>Relocate the gate so flow occurs in only one direction.</p> <p>Increase the process temperature. Reduce the pack pressure. Increase the mold temperature.</p>
<p>SURFACE DEFECTS Uneven shine Beach marks</p>	<p>High molecular weight formulation in highly polished mold.</p>	<p>Change to a lower molecular weight formulation. Texture mold cavity surface (EDM, sandblast, etc.)</p>
<p>SURFACE DEFECTS Silver streaking or splay marks</p>	<p>Contaminated material.</p> <p>High shear in the material.</p>	<p>Check for moisture (condensation) or dry the material. Check regrind for contaminants or moisture. Dry if required Reduce injection speed. Increase the process temperature. Increase the gate size.</p>

Injection Molding Troubleshooting

STARTUP ISSUES (CONT.)

OBSERVATION	CAUSE	SOLUTION
EJECTOR PIN MARKS	Parts soft during ejection.	Increase mold close time. Reduce mold temperature. Reduce process temperature. Texture mold surface for better release. Increase size of pins. Increase draft on part. Reduce wall section. Use a formulation with mold release. Use a harder formulation.
BURNT SMELL Parts have a yellow cast	Degraded material.	Purge machine and observe whether problem reoccurs. Reduce process temperature. (Particularly in rear of machine). Reduce regrind level. Reduce residence time. Purge machine after shutdown. Move to a smaller machine. Reduce hot runner system temperatures. Minimize dead spots in hot runner manifold.
PART STICKS IN "A" HALF or STATIONARY SIDE OF THE TOOL	Insufficient extraction force.	Sandblast "A" side of tool. Polish "B" side. Run "A" side cooler. Put keepers in "B" half of tool. Increase draft on part in "A" half of tool.
PART STICKS DURING EJECTION	Insufficient ejection force.	Increase mold closed time. Reduce mold temperature. Reduce process temperature. Reduce pack and hold pressure. Sandblast "B" side. Increase size of ejector pins. Increase number of ejector pins. Increase draft on part. Provide air assist.
NON-UNIFORM COLOR	Poor dispersion. Contamination.	Increase back pressure and/or screw rpm. Change the color concentrate carrier to a material with a lower melt temperature. Check for clean regrind.

Index

- Anisotropy, 5
- Back pressure, 25
- Barrel capacity, 22
- Barrel temperatures, 24
- Clamp capacity, 21
- Cold runner design, 11–13
- Cold slug wells, 12
- Color concentrate carriers, 22–23
- Coloring, 22–23
- Cooling time, 26
- Cushion, 25, 26
- Degradation, 27
- Diaphragm gates, 15
- Direct gates. *See* Sprue gates
- Draft, 17
- Drying, 22
- Edge gates. *See* Tab gates
- Ejector pins, 17
- Extended nozzles, 8
- Fan gates, 14
- Flow length, 4
- Flow to thickness ratio, 4
- Flow vs. cross-flow direction, 5
- Gate design, 13–15
- Gate location, 15
- Grinding guidelines. *See* Regrind
- Hard formulations, 28
- Hold time, 26
- Hot runner systems, 17–20
- Hot sprue bushings, 8
- Hot tip gates, 19–20
- Injection pressure, 26
- Injection speed, 25
- Injection time, 26
- Machine selection, 21
- Manifold design, 18
- Mold cooling, 17
- Mold finish, 7
- Mold surface, 7
- Mold temperatures, 24
- Mold texture, 7
- Mold types, 6
- Nozzle selection, 22
- Overmolding, 20
- Overpacking, 27
- Part ejection, 17
- Purging, 23
- Radii/fillets, 4
- Regrind, 23
- Runner balance, 10–11
- Runner keepers, 13
- Screw position, 26
- Screw rpm, 25
- Screw selection, 22
- Shear dependence, 3
- Shear rate, effect on viscosity, 3
- Shot size, 25
- Shrinkage, 5
- Soft formulations, 27
- Spiral flow, 4
- Sprue design, 8
- Sprue gates, 15
- Sprue puller design, 8–10
- Steel selection, 6–7
- Submarine gates, 14
- Sucker pins. *See* Runner keepers
- Tab gates, 13
- Transfer pressure, 26
- Troubleshooting, 28
- Tunnel gates. *See* Submarine gates
- Undercuts, 5
- Underpacking, 27
- Valve gates, 18–19
- Vent size and depth, 16
- Venting, 16
- Wall thickness, 4

1.844.4AVIENT
www.avient.com



Copyright © 2020, Avient Corporation. Avient makes no representations, guarantees, or warranties of any kind with respect to the information contained in this document about its accuracy, suitability for particular applications, or the results obtained or obtainable using the information. Some of the information arises from laboratory work with small-scale equipment which may not provide a reliable indication of performance or properties obtained or obtainable on larger-scale equipment. Values reported as "typical" or stated without a range do not state minimum or maximum properties; consult your sales representative for property ranges and min/max specifications. Processing conditions can cause material properties to shift from the values stated in the information. Avient makes no warranties or guarantees respecting suitability of either Avient's products or the information for your process or end-use application. You have the responsibility to conduct full-scale end-product performance testing to determine suitability in your application, and you assume all risk and liability arising from your use of the information and/or use or handling of any product. AVIENT MAKES NO WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, either with respect to the information or products reflected by the information. This literature shall NOT operate as permission, recommendation, or inducement to practice any patented invention without permission of the patent owner.