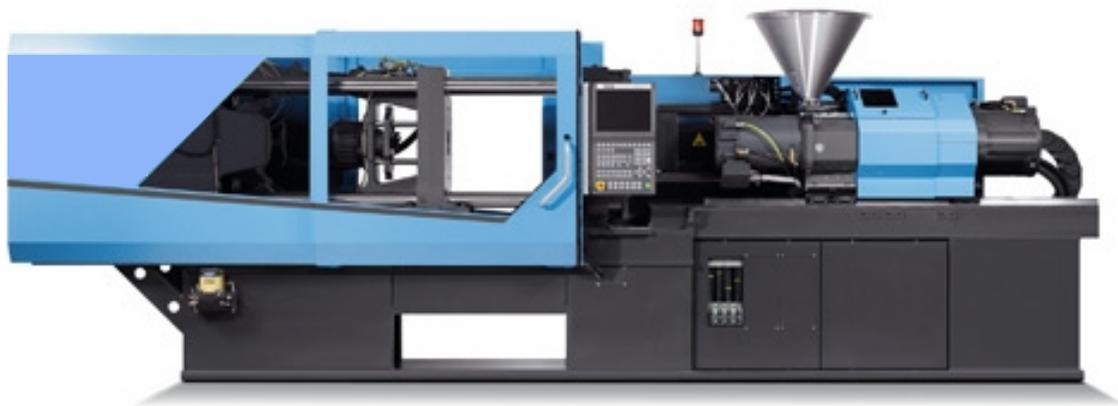




ESPOLEX®

THERMOPLASTIC ELASTOMER

Injection Molding Design & Processing Guide



This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using this information. Sumitomo Chemical Co., Ltd. Assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.

Sumitomo Chemical **North America**

Sumitomo Chemical **Europe**

Sumitomo Chemical **Asia**



Table of Contents

◆ Introduction.....	3
◆ General Handling.....	4
○ Standard Packaging	
○ Drying	
○ Purging the Barrel	
○ Risks and Precautions	
◆ Equipment/Tooling Choices.....	5
○ Molds/Tooling	
○ Molding Machines	
◆ Part and Runner Design.....	9
○ Part Design	
○ Runners	
◆ Gate Design.....	14
◆ Tool Design.....	15
○ Vents	
○ Ejection Mechanisms	
○ Reducing the Severity of “Sticking”	
◆ Processing Guidelines.....	18
○ Temperature	
○ Pressure	
○ Velocity	
○ Time	
◆ Properties of ESPOLEX®	26
◆ Troubleshooting.....	27
◆ References.....	29

Introduction

The ESPOLEX® TPE product portfolio includes olefin based Thermoplastic Elastomer (TPE) and Thermoplastic Vulcanizate (TPV) materials with a broad range of hardness, flexural modulus, and flow-ability. Each product was originally designed with a particular application in mind, and a goal to design that product as a technology leader of that particular application. With this in mind, you can understand that these products are unique, and therefore demand unique product and tool design standards. This guide will address those attributes of product and tool design that can either broaden or restrict the “processing window”.

Please be aware that following these guidelines will not guarantee that the product made will be free of defects. Many variables exist in the manufacturing process, and it is very difficult to understand the effect that each variable will have on the end product. This design guide should only be used as basic building blocks. The final design may need to be tweaked to create the desired end product.

General Handling

◆ STANDARD PACKAGING

Standard packaging for ESPOLEX® TPE materials comes in 1,500 pound (680 kg.) Gaylords for resins produced in North America, and 50 pound (23 kg.) bags for resins produced in Asia. If another style of packaging is needed, please contact your sales representative.

◆ DRYING

ESPOLEX® TPE is non-hygroscopic, and drying is therefore only necessary if condensation is visible on the pellet surface or on the inside of the resin liner. If drying is necessary, dry for 2-4 hours at 180° F (82° C).

◆ PURGING THE BARREL

Before molding new material grades or colors, it is highly recommended to purge the remaining material out of the barrel first. To purge out the remaining resin, add at least two pounds of purging compound into the hopper. Continue to purge material out of the nozzle (purge thru the hot drop, when using hot runner systems) until there are no signs of the remaining material exiting the nozzle. Once this is complete, the new material grade or color can be added to the hopper, and process set-up for the new product can commence.

◆ RISKS & PRECAUTIONS

For safety information, please review the Material Safety Data Sheet (MSDS) for the specific ESPOLEX® grade. Do not open the protective liner, until the resin is ready for use. Once the liner is opened, the material is highly vulnerable to contamination, so it is important to make sure the container is properly sealed so that airborne debris and/or spilled materials cannot contaminate the resin.

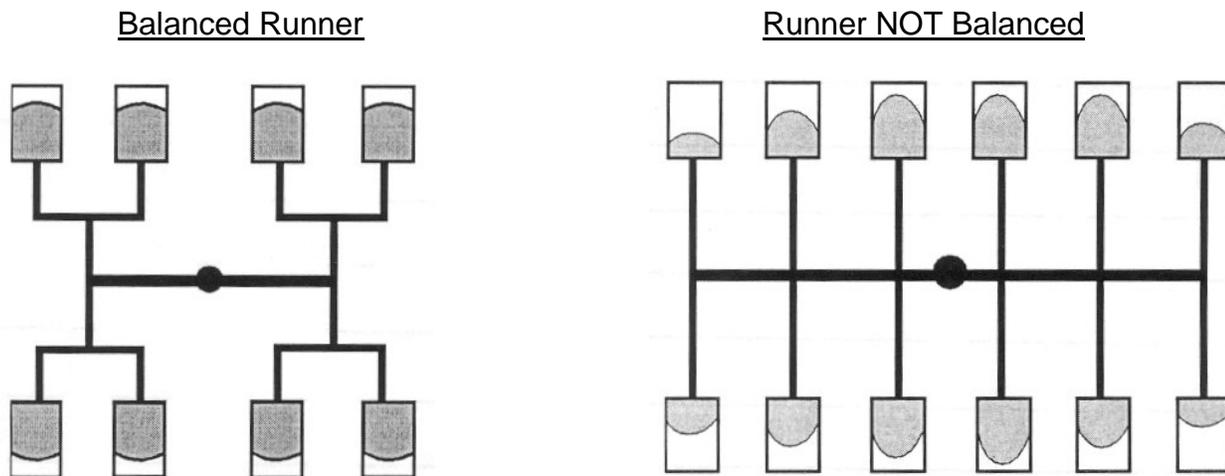


Equipment/Tooling Choices

MOLDS/TOOLING

Multiple Cavities:

Single cavity tools should be used when possible. If a multiple cavity tool is necessary, the cavities, runners, and gates must be balanced. Also make sure that the injection molding press is capable of exceeding the pressure needed to properly fill all cavities.



*Illustration from "RJG Technician Training & Master Molder Certification Program Master Molder 1" binder.

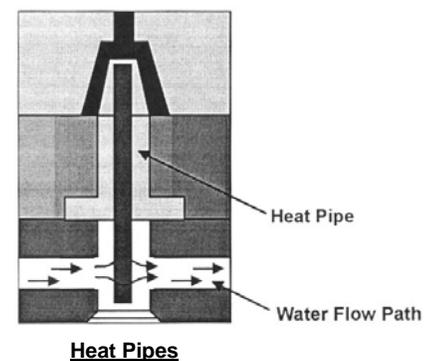
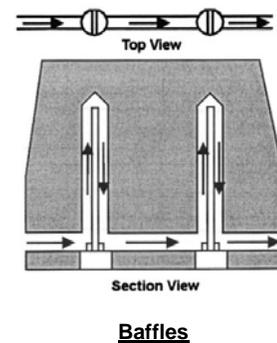
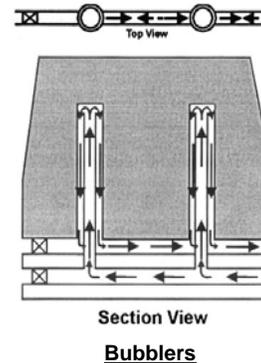
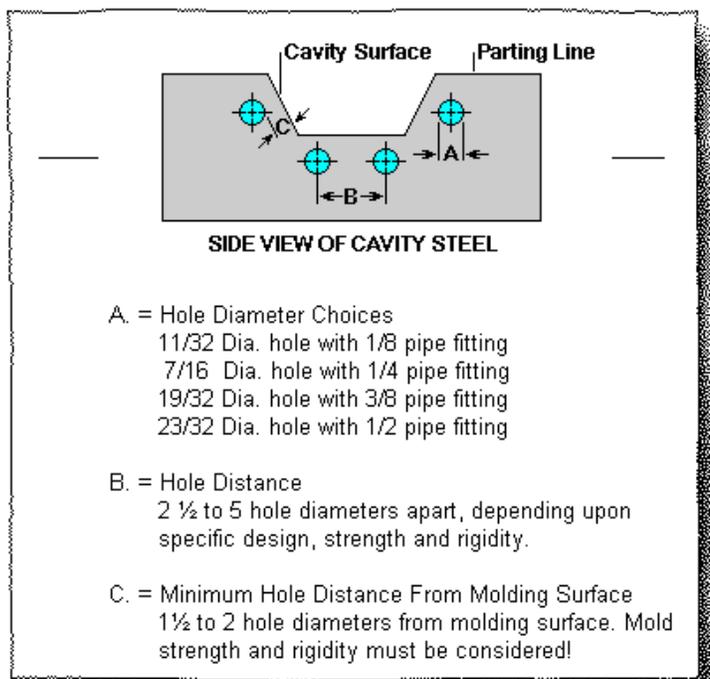
Cold runner systems are recommended in multiple cavity tools. Hot runner system can lead to uneven and inconsistent filling of the cavities. Minor fluctuations in temperature from one hot tip zone to another can create fill inconsistencies within minutes. Fill inconsistencies can also occur when a cold slug develops in one hot tip, but not in the others. Depending upon which cavity fills first, the cold slug can move from one hot tip to another causing inconsistent fill patterns.



Equipment/Tooling Choices

Coolant Lines:

For complete cooling channel design, consult your tool maker. The basic rule of thumb for cooling channel design is to choose a diameter (D); larger diameters remove more heat; place the cooling channel a distance of 1.5 - 2.0 times D from the surface of the mold; Pitch, the distance between each cooling channel, should be 2.5 - 5 times D. Make sure all cooling channels are equal distance from the cavity and core surfaces. Bubblers, Baffles, and Heat Pipes can be used to remove heat from the hard to reach cores. Materials such as Copper or Aluminum dissipate more heat than steel.

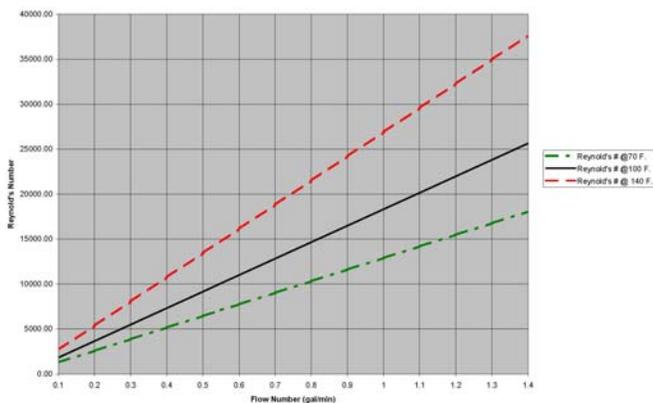


*Illustrations from "RJG Technician Training & Master Molder Certification Program Master Molder 1" binder.

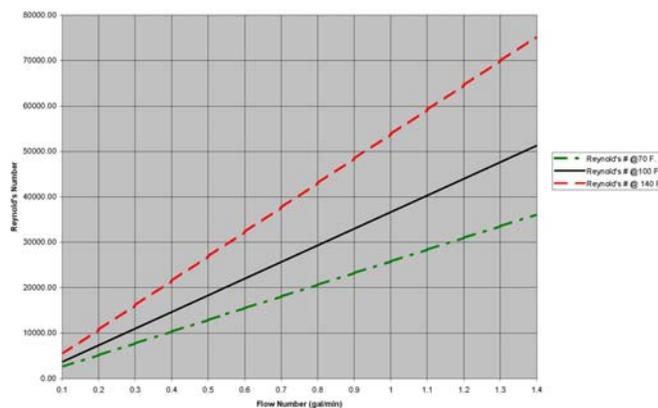
Equipment/Tooling Choices

Laminar flow of coolant in a cooling channel is when the coolant flows smooth without disruption. Turbulent flow of coolant in a cooling channel is when the coolant tumbles through the cooling channel in a random manner. Turbulent flow of the cooling fluid removes more heat from the steel than laminar flow, and is preferred for optimum cooling. To reach turbulent flow, a specific flow rate must be reached. This flow rate is dependent upon the diameter of the cooling channel. To determine if your coolant is experiencing turbulent flow, measure the flow rate with a flow meter, and use the charts below to determine the Reynold's number. Turbulent flow has a Reynold's number of 5,000 or greater.

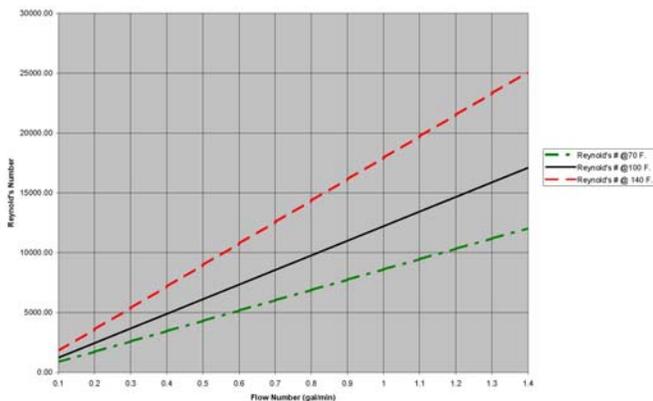
Reynold's Number based on Flow Number for a 1/4" inside pipe diameter.



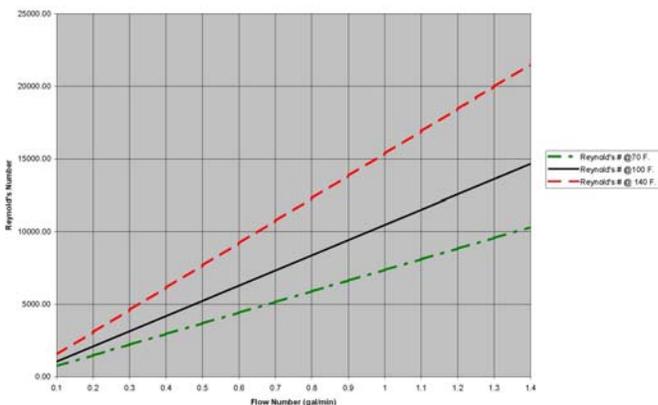
Reynold's Number based on Flow Number for a 1/8" inside pipe diameter.



Reynold's Number based on Flow Number for a 3/8" inside pipe diameter.



Reynold's Number based on Flow Number for a 7/16" inside pipe diameter.



This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using this information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.



Equipment/Tooling Choices

◆ MOLDING MACHINES

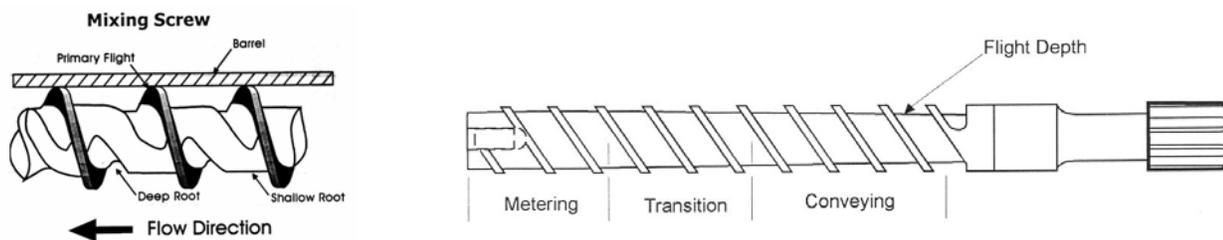
Molding machines with closed loop process control are necessary to produce consistent products. Regular preventive maintenance should be initiated to ensure the equipment is working properly.

Barrels:

Appropriately sized barrels are necessary to ensure the resin is melted, blended, but not degraded. Make sure the shot size is between 30% and 80% of the total barrel size.

Screws and Nozzles:

Screws can have a considerable effect on the melting and blending of your resin. When molding with pre-colored resins, a general purpose screw with an intensification ratio of 2:1 should be sufficient. But when processing a natural resin blended with a color concentrate, a salt and pepper product, a mixing screw, a high intensification ratio screw, and/or a mixing nozzle may be necessary. If you find it necessary to increase the back pressure and/or screw rpm above the material manufacturer's recommended settings (listed under the "Processing Guidelines" section) in order to improve mixing, then you need to install a different screw and/or nozzle design. When in this situation, it is best to ask your screw manufacturer for the proper screw design. Nozzles should be drilled out to match a diameter of 10% less than the sprue "O" diameter



Machine Tonnage (MT):

A machine tonnage of 2.5-3.0 metric tons per square inch should be sufficient for most parts. The required tonnage can be determined by taking the projected surface area of your part (PPA) in square inches times the number of cavities (TC) plus the projected surface area of your runner system (RPA) in square inches times the multiplication ratio (MR) of 2.5-3.0.

$$MT = ((PPA \times TC) + RPA) \times MR$$

This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using this information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.

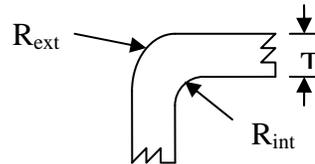
Part & Runner Design

◆ PART DESIGN

When considering total part design, the actual part design is as important as the runner/gate design. Part design is determined by many considerations including the desired location and size of its attachment components, the environment that the part will be subjected to in the field, and the desired styling. Although we may not have much control over the design of these characteristics, we can follow some standard guidelines to ensure that design pitfalls are avoided, and the runner and gate are designed appropriately.

Radiuses:

To avoid crack propagation, all corners (inside and outside) need to include a radius.



The internal radius of a corner should be approximately half of the wall thickness of the adjoining sections.

$$R_{INT} = T/2 \text{ +/- } 10\%$$

The external radius of a corner should be the wall thickness plus the internal radius.

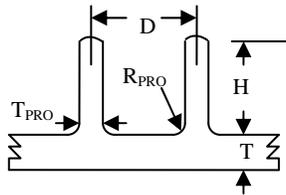
$$R_{EXT} = (R_{INT} + T) \text{ +/- } 10\%$$



Part & Runner Design

Protrusions (Ribs and Bosses):

Ribs and bosses that are excessively thick can cause sink on the opposite side of the part and/or voids. To avoid this effect, be sure to follow proper design guidelines when determining the thickness of protrusions.



The wall thickness at the base of all protrusions should be between 50% and 60% of the nominal wall thickness.

$$0.5T \leq T_{\text{PRO}} \leq 0.6T$$

The radius at the base of a protrusion should be between 25% and 50% of the nominal wall thickness.

$$0.25T \leq R_{\text{PRO}} \leq 0.5T$$

To avoid difficulty during ejection, all protrusions should have no less than 2° of draft per side with an optimum draft of 4° per side, and a height of no more than five times the wall thickness. Even after adding draft, make sure that the base of the protrusion follows the above guidelines.

$$H \leq 5T$$

To allow the steel to offer proper heat transfer during molding, be sure to properly separate each protrusion from one another. A good rule of thumb is to keep a distance of no less than three times the height of the protrusions.

$$D \geq 3H$$

Part & Runner Design

Over-Molding Design:

The TPE section should maintain thicknesses between 1.0 – 3.0, mm for good adhesion to the substrate, but make sure thickness transitions do not exceed 20%.

TPE materials stick to polished steel, so adding texture can improve de-moldability. Be sure to keep the surface of the substrate, where the TPE will adhere, polished so that the adhesion is strong. Adding texture to the runners and gates, may also lower your cycle time.

The wall thickness of the TPE section should be less than or equal to the wall thickness of the hard substrate to avoid the differences in shrinkage from causing warpage.

Flow lengths can affect bond strength, so make sure to maintain an L/T (length-of-flow/wall thickness) ratio of less than 150:1 for the TPE section. If this ratio must be exceeded, multiple gates should be used.

Part & Runner Design

Nominal Wall Thickness

Wall thicknesses should be no more than 5 mm and no less than 1 mm, targeting at least 2% of the longest flow length. To avoid warpage, limit wall thickness transitions to 15% increases or 15% decreases. An even wall thickness throughout the part, will help ensure weld-lines do not form at the end-of-fill.

Soft Touch Designs

When looking for a certain soft feel, thickness can play an important role. Thicker TPE sections can feel softer when over-molded onto a harder substrate. Although when not over-molding, thinner TPE sections can feel softer depending upon the material's Flexural Modulus. Lower Flex Mod materials will feel softer, whereas higher Flex Mod materials will feel harder/stiffer. Adding texture can also make the part feel softer.

Tear Seams

Tear seams should be no less than 0.5 mm thick, and have a tear seam radius of at least 60°. Try to position the gate so that it is of equal distances from each section of the tear seam.

Attachment Types & Holes

Many attachment types can be used. When choosing the attachment type and quantity of attachment points, be sure to confirm that the design can absorb the force exhibited during the part's use. Holes and/or attachment types will cause a weld-line to form. Great care during the design phase is needed to ensure the weld-line is located in an area that will experience little or no force during the part's use. Placing a gate near the hole or attachment point can also increase the strength of the weld-line.

Part & Runner Design

RUNNERS

Runner Type:

Full round runners are preferred, since they create the least resistance during the injection phase. If another type of runner is necessary, a half round runner would be the second choice, with the trapezoidal runner being the third.



Runner Length:

Runner length should be limited as much as possible. The shorter the runner length, the less resistance, and the better transfer of pressure from the screw to the part.

Cold Runners:

To size the runners appropriately, The sub-runner should be 1.5 times the thickest section of the part (WT). The main runner should be 2.5 times WT, the sprue “O” diameter should be 3.5 times WT, and the base of the sprue should be no more than 4.5 times WT. The draft of the sprue should be at least 2° per side. At a minimum, there should be at least a 20% increase in thickness around each turn from the runner closest to the gate till the main runner is reached.

Hot Runners:

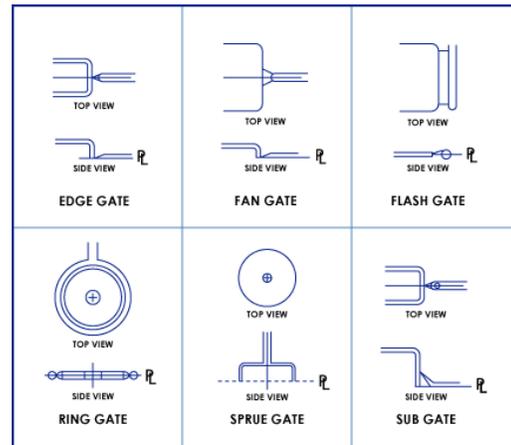
Full round gates are necessary for hot runner systems, but the diameter can be less than the required diameter for cold runner gates. A diameter of a hot runner gate can be 35% of the nominal wall thickness. When using a hot runner, it is essential to install valve gates for each cavity. The molten resin can drool into the mold when a valve gate is not used, causing surface defects. Hot runners designed without a valve gate can also allow a cold slug to form prior to injection. The cold slug can be seen as a speck, streak, or splay on the parts surface. If a hot drop is used to direct material into a cold sprue, the hot drop should be no less than 35% of the thickest section of the sprue. Hot runner systems are not recommended for un-painted class A appearance parts, due to the potential unevenness in material temperature which can negatively effect the surface appearance.



Gate Design

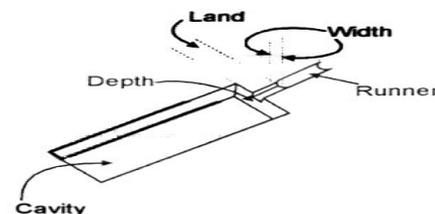
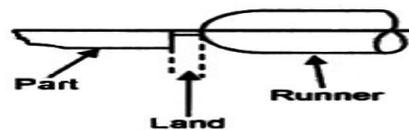
Gates:

Different gate designs can be seen in the diagram on the right. Fan gates or edge (standard) gates are preferred in any application with the depth being about 50% - 90% of the thickest section of the part (typically 1-2 mm), the width being at least twice the depth (typically about 5 mm for parts less than 200 cm³ and about 8 mm for parts greater than 200 cm³), and the land not to exceed 0.77 mm.



*Illustrations from "RJG Technician Training & Master Molder Certification Program Master Molder 1" binder.

Sub/Cashew/Tunnel Gates can also be used when necessary. For this gate style, the gate diameter should be between 60% and 90% of the nominal wall thickness depending upon the flow length. It is best to inject against a surface so as to reduce the possibility of jetting. Due to the increase in shear, these types of gates can cause gate blush and should be avoided in un-painted class A appearance parts.



Gate location is best determined after running a mold flow analysis. If mold flow analysis is not readily available, there are some guidelines that can be followed. The gate should be located at the thickest section of the part, with the end-of-fill being the thinnest section of the part. Make sure not to gate into a thin section of the part. When choosing the gate location on an airbag cover, it is important to understand the type of tear seam being used. When using an H-shaped tear seam, place the gate at the 12 or 6 o'clock positions. With an I-shaped tear seam, place the gate at the 3 or 9 o'clock positions. With a U-shaped tear seam, a center gate should give you the best results.

Tool Design

VENTS

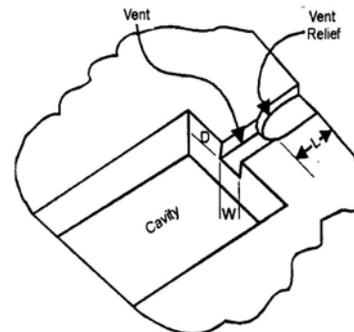
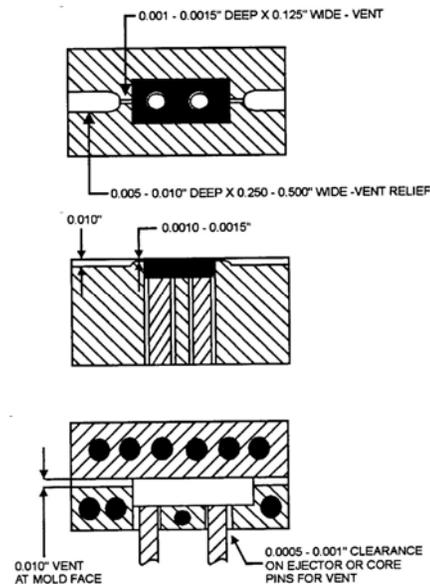
Gas entrapment can cause many problems from burns to weak weld-lines, to uneven gloss on the surface. It is essential to ensure that each cavity is properly vented, to avoid gas entrapment and burns.

Part Vents:

Vents should be located at a minimum 1 for every parting line inch, with a target of having full venting along the perimeter of the parting line. Vents should be 5 mm wide and 0.013 - 0.025 mm deep. Be sure to start with a small vent depth and increase as needed.

Runner Vents:

Runners should also be vented to help evacuate gasses. Runner vents should be the same width as the diameter of the runner being vented. The depth should be about 0.05 mm. Go out 1.5 mm from the runner at these dimensions, and drop into a 1 mm channel to atmosphere. Always use self-cleaning vents at the end of all core pins. This will help avoid unintended knit lines.



*Illustrations from "R/JG Technician Training & Master Molder Certification Program Master Molder 1" binder.



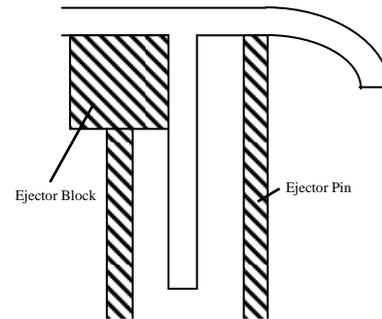
Tool Design

EJECTION MECHANISMS

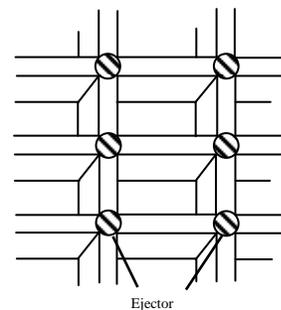
Proper ejection is extremely important in order to remove the part from the tool in a timely manner without damaging the part. Multiple ejection types can be used, but generally the more ejection square area, the better results you will have.

When placing ejection mechanisms near ribs and protrusions, always place the ejector or blade on the rib or protrusion, and never place them inside or next to the rib or protrusion. For ribs, the ejector pin or blade should be placed at the intersection of the ribs.

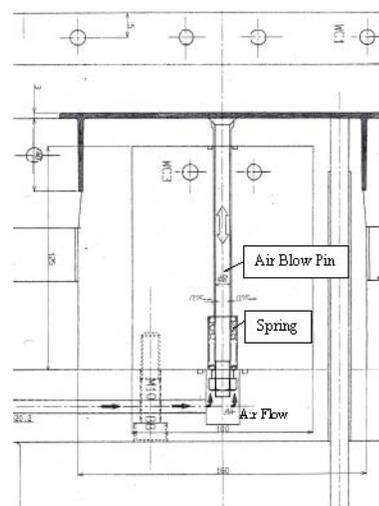
Air poppets are a helpful mechanism when it comes to ejecting parts. The air poppets will help eliminate any vacuum between the part and the tool surface, and will help push the part off of cores and lifters.



Ejector Examples



Ejector location for Ribs



Air Poppet Design

This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.



Tool Design

◆ REDUCING THE SEVERITY OF “STICKING”

TPE materials contain a high content of rubber, which can contribute to the potential for parts “sticking” in the tool. The severity of “sticking” can be reduced, by following some basic design guidelines.

The first step is to add ample draft in all areas of the part that are in line with the mold open direction. A draft of 4° per side is preferred, but a draft of at least 2° per side is necessary.

Proper cooling in the tool, especially in areas containing ribs and protrusions, is necessary to reduce “sticking” (see pages 6 & 7).

Proper use of ejection mechanisms is also necessary to reduce “sticking” (see page 16). Generally the more square area of ejection used, the better your results of removing the part from the tool.

Adding a texture to the tool will also help reduce “sticking”, by eliminating the suction cup effect. Rubber tends to squeeze out air and cause suction to the steel surface, but when a texture is added air pockets at the surface will disrupt this suction cup effect. A light bead blast may be all that is needed.

After all items listed above are addressed, the last step is to add a coating to the surface of the steel. Typical coatings include Nickel/Teflon and Electroless Nickel Boron Nitride. The A-side of the tool should always be coated. The B-side can either be completely coated or the coating can be targeted for specific areas of concern.



Processing Guidelines

ESPOLEX® TPE Product	Melt Temp. in °F (°C)		Mold Temp. in °F (°C)		Injection Velocity
	Temp min	Temp max	Temp min	Temp max	
<i>Standard Settings</i>	390 (200)	470 (245)	75 (25)	120 (50)	-High (when using low melt temperatures)
<i>Settings for improved adhesion to substrate</i>	420 (215)	490 (255)	95 (35)	130 (55)	-Med (when using high melt temperatures)

ESPOLEX® TPE Product	Injection Pressure	Back Pressure (psi)	Screw RPM
<i>Standard Settings</i>	20% more than the pressure required to meet the desired injection velocity	50 - 145 (blends: 145-200)	Only fast enough to recover the screw 3 seconds before the mold opens.

This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.

Processing Guidelines

TEMPERATURE

Barrel Temperature

In general, raising the barrel temperatures improves flow-ability, due to the reduced viscosity of the molten material, which may improve part appearance. However, excessively high temperatures might cause thermal degradation of the material and / or de-lamination within the molded part. Excessively low temperatures will decrease the flow-ability of the material, which may make it difficult to obtain a good surface appearance. Make sure to check the previous page for the recommended melt temperature.

Mold Temperature

In general, higher mold temperatures may improve the part appearance, however, cycle time for molding may increase. If de-lamination (peeling) on the surface of the part occurs at high mold temperatures, the mold temperature should be lowered. If the nominal wall thickness of the part is thin, such as less than 2 mm, the mold temperature should be at the high limit of the suggested conditions listed on the previous page. Temperature of the core half of the mold should be equal to that of the cavity half.



Processing Guidelines

PRESSURE

Understanding the Relationship between Injection Pressure and Injection Velocity

A pump makes oil flow, but there must be resistance to flow to create pressure. We cause things to move in the injection molding process, including plastic, by controlling oil flow, not pressure. The pressure results from the resistance to the flow of plastic. Pressure can provide information about the process. The injection pressure setting, in an injection molding machine, is a parameter used to limit the oil flow for safety purposes.

The molding machine must have a high enough injection pressure setting to be able to meet the velocity profile set by the technician.

The molding machine will only use the oil flow it needs to reach the set velocity. Any excess oil flow will not be used and will be redirected back to the reservoir. The resin is constantly experiencing fluctuating levels of resistance as it flows into the mold. To compensate for this fluctuating level of resistance, the machine will pull from its reservoir of oil to maintain the flow it needs so that it can maintain the velocity set by the technician.

In such a case as when the injection pressure is set too low, the injection velocity will fluctuate. As the resin experiences resistance to flow, the machine will try to pull more oil from its reservoir to compensate for the excess pressure, but the machine will not be able to compensate for the excess pressure because the injection pressure setting is set too low, forcing the molding machine to redirect the oil back to the reservoir. The injection velocity will therefore fluctuate directly with fluctuations in the resistance to flow.

The quality, and surface appearance, of the part being molded is greatly affected by injection velocity. Therefore, it is very important to be able to control the injection velocity instead of allowing the tool design to control it for you.

The bottom line: To consistently meet a desired injection velocity, the injection pressure must be set higher than what is needed.

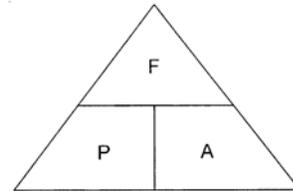


Processing Guidelines

Setting Injection Pressure

During the initial start-up, the injection pressure should be set to 95% of the maximum allowable. Once the maximum injection pressure needed by the molding machine to mold a quality part (injection pressure at transfer) is determined, the maximum injection pressure setting should be determined by using the following equation:

Force = pressure times area
Area = force divided by pressure
Pressure = force divided by area



*Illustration from "RJG Technician Training & Master Molder Certification Program Master Molder 1" binder.

Maximum Injection Pressure Setting = (1.2) X (Injection Pressure at Transfer)

This method should allow enough injection pressure to maintain a constant injection velocity, and still include a safety factor in case the molten material is subjected to extreme resistance.

Processing Guidelines

Holding Pressure

Holding pressure varies according to the design of the mold and gate. It is necessary to determine the optimum holding pressure. If holding pressure is too high, it may cause flash and/or warp. If holding pressure is too low, sink marks may be observed. Therefore, it is necessary to find a well-balanced point. To set the initial holding pressure, refer to the following equation:

$$\text{Holding pressure} = \text{Injection pressure at transfer} \times (0.3 \text{ to } 0.6)$$

Back Pressure

Back pressure is recommended to be set:

-  Low, less than 1 MPa (145 psi), for pre-colored resin
-  High, more than 1 MPa (145 psi) and less than 1.4 MPa (200 psi), for "salt & pepper".

In the case of pre-colored resin, if the back pressure is set too high, the material may degrade. In the case of "salt & pepper", too low of a back pressure may cause insufficient mixing of the colorant and base resin.

Processing Guidelines

VELOCITY

Injection Velocity

- a) When the barrel temperature is set at the low half of the recommended settings, the injection velocity should be set near the high limit to get a good part appearance. (Injection rate should be more than 100 g/sec.)

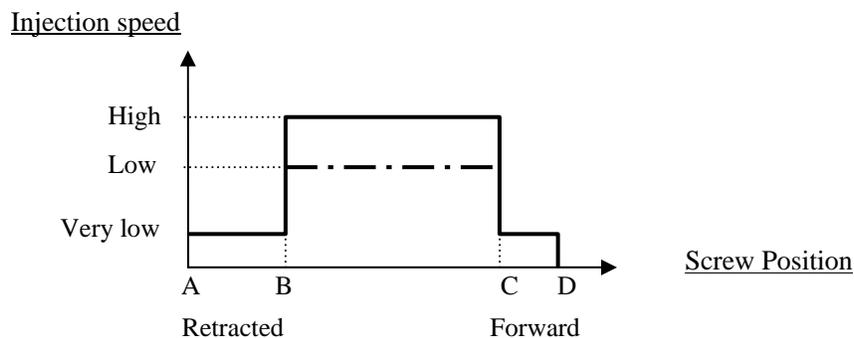
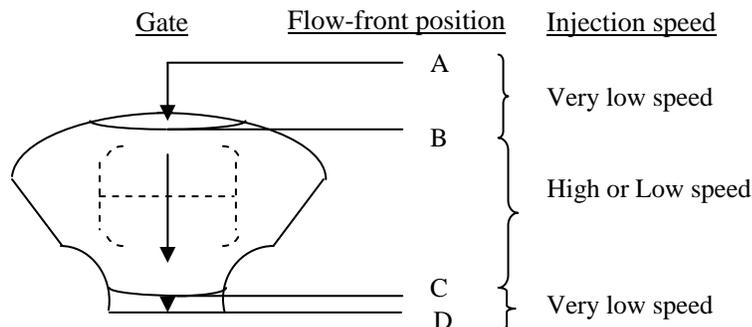
- b) When the barrel temperature is set at the high half of the recommended settings, the injection velocity should be set near the low limit to get a good part appearance. (Injection rate should be less than 70 g/sec.)

Method (a) is typically better than method (b) to obtain a good part appearance, since shear has more influence on flow-ability than temperature. If shear related defects are observed, such as de-lamination, weak weld-lines, and streaks, use method (b).

Processing Guidelines

Setting Injection Velocity (Standard Settings)

To avoid jetting, injection speed may need to be very slow when the molten resin is initially flowing through the gate. Once the flow front of the molten resin is through the gate, the injection speed may be increased depending on the barrel temperature settings (see previous page for details) and kept just prior to fully packing the part. In the final stage, the injection speed is decreased to allow air to escape from the mold, and the slow speed is maintained until the part is 95% full. If the injection velocity is too high at the end of fill, burning or trapped air may occur



Processing Guidelines

TIME

Holding Time

Holding time is dependent on the time it takes for the gate to freeze, and varies according to the barrel temperature, mold temperature and design of the gate. When using a cold runner, hold time should start short and slowly be increased. After each increase in hold time, each part should be weighed until the part weight ceases to increase. When using a hot runner, hold time should start short and be slowly increased until the optimum part dimensions are reached. Too long of a holding time may cause flash and / or warp. Too short of a holding time may lead to sink marks. Therefore, appropriate holding time should be determined.

Cooling Time

Cooling time varies widely, according to the barrel temperature, mold temperature and thickness of a part. Longer cooling times can prevent sink marks and part deformation; however, they will also increase the cycle time. Shorter cooling times may allow for insufficient in-mold shrinkage, which may cause sink marks and part deformation, but will also decrease the cycle time. Therefore, it is necessary to determine the optimum cooling time.

A cooling time estimate can be determined by taking the maximum part thickness in millimeters and multiplying by 10.



Product Properties

Injection molding grade

Item		Test Method and Conditioning	Units	High performance grade				Standard grade			
				Flexible		Semi-rigid		Semi rigid	Rigid		
				3675	3785	3885	3255 Black	901	907	903	
Physical property	Specific gravity	ISO 1183	kg/m ³	880	880	880	880	900	900	910	
	Melt flow rate (MFR)	ISO 1133 21.18N	g/10min	-	-	1.5	20	8	7	5	
		ISO 1133 98.07N	g/10 min	30	50	>100	>100	-	-	-	
Mechanical property	Durometer A Hardness		ISO 868	-	60	70	85	95	-	-	-
	Durometer D Hardness			-	-	-	-	49	41	52	60
	Flexural modulus		ISO 178	MPa	-	-	65	200	170	350	550
	Tensile strength	100% modulus	ISO 37 Type 1A 500mm/min	MPa	1.8	2.6	3.5	YS 5.9	YS 8.1	YS 11	YS 17
		Braking Strength		MPa	4.6	5.5	9.3	20	15	15	20
		Elongation at break		%	580	530	650	700	620	700	640
Impact strength	Notched Izot Impact at 23°C	ISO 179 3.2mmT	kJ/m ²	NB	NB	NB	NB	NB	NB	NB	
	Notched Izot Impact at -30°C			NB	NB	NB	NB	5	15	54	
Thermal property	Brittleness temperature		ISO 812 Type A	°C	<-60	<-60	<-60	<-60	<-60	<-60	
Others	Compression set		ISO 37 Type A 23°C 22hrs	%	27	27	-	-	-	-	
			ISO 37 Type A 70°C 22hrs	%	35	35	-	-	-	-	
Major Application areas			<ul style="list-style-type: none"> ••Automotive interior (Packing grid etc.) ••Automobile exterior (mold etc.) ••Gasket ••Packing ••Bathroom articles ••Mat etc. 	<ul style="list-style-type: none"> ••Automobile exterior (mole, Mud guard, Weather stripping etc.) 	<ul style="list-style-type: none"> ••Automobile exterior (Mud guard, Step mat etc.) ••Recreational equipments ••Knobs 						

Typical Injection condition of SUMITOMO TPE

1. Cylinder temperature
 - Bottom: 180-200°C
 - Center: 200-220°C
 - Top: 210-230°C
 - Nozzle: 210-230°C
2. Mold temperature: 40-60°C
3. Injection Speed: Faster

- 1) The values given are typical averages and not to be considered as sales specification limits or guaranteed values.
 - 2) Unless otherwise specified, non-rigid grade test specimens are compression molded while semi-rigid and rigid grade test specimens are injection molded.
- All tests are conducted at 23°C.

This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using this information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.



Troubleshooting Guide

	Pressures				Speeds				Positions				Temperatures				Times				Other																					
	Increase Injection Pressure Setting	Increase Pack/Hold Pressure	Decrease Pack/Hold Pressure	Increase Back Pressure	Decrease Back Pressure	Increase Clamp Tonnage	Decrease Clamp Tonnage	Increase Injection Velocity	Decrease Injection Velocity	Use Profile Injection to Slow Down	Increase Screw RPM	Decrease Screw RPM	Increase Clamp Breakaway Speed	Decrease Clamp Breakaway Speed	Decrease Ejection Velocity	Increase Decompression	Decrease Decompression	Add Sprue Break	Remove Sprue Break	Increase Cushion or Shot Size	Decrease Cushion or Shot Size	Increase Nozzle or Hot Runner	Decrease Nozzle or Hot Runner	Increase Melt Temperature	Decrease Melt Temperature	Increase Mold Temperature	Decrease Mold Temperature	Adjust Temperature of Mold Halves	Increase Residence Time	Decrease Residence Time	Increase Pack/Hold Time	Decrease Pack/Hold Time	Increase Cooling Time	Decrease Cooling Time	Decrease Mold Open Time	Clean/increase Venting	Check for Contamination	Purge Barrel if Contamination Present	Check for Proper Gate/Runner	Use Mixing Nozzle or Screw		
Drop																																										
Nozzle/Tip Frozen																																										
Cold Slugs																																										
Sticking of Parts																																										
Sprue (sticking or breaking)																																										
Ejector Marks/Holes (Pin-Push)																																										
Blush/Gate																																										
Blemish/Smears																																										
Bubbles/Blisters																																										
Burns/Dieseling/Black Surface																																										
Black Specks (Dark Spots)																																										
Poor Color Distribution																																										
Black/Brown/White Streaks																																										
Cloudy or Hazy (clear parts)																																										
Crazing, Stress Whitening																																										
Delamination																																										
Gloss: Low or High																																										
Flash																																										
Short Shots																																										
Sinks																																										
Flow Lines																																										
Grooves/Ripples																																										
J-Hooks																																										
Jetting																																										
Splay/Silver Streaks																																										
Strings/Stringers / Wisps																																										
Non-melts/Windows																																										
Warpage																																										
Weld-lines																																										
Brittleness, Stress Cracks																																										
Dimensions Unstable / Inconsistent Parts																																										
Voids																																										

**Pressure Limited is a situation where the injection pressure setting is less than the maximum injection pressure required by the machine to reach the desired injection velocity. This can cause many different processing defects. If this is the case, the injection pressure setting must be increased.

■ Items highlighted in red may reduce your profit

This document reports accurate and reliable information to the best of our knowledge, but our suggestions and recommendations cannot be guaranteed because the conditions of use are beyond our control. Information presented herein is given without reference to any patent questions which may be encountered in the use thereof. Such questions should be investigated by those using this information. Sumitomo Chemical Co., Ltd. assumes no responsibility for the use of information presented herein and hereby disclaims all liability in regard to such use.



Troubleshooting Guide

❖ The troubleshooting guide didn't solve your problem?

For more detailed guidance on troubleshooting, reference the “SPE Plastics Technician’s Toolbox” books 6 & 7 {call 1-(203)775-0471 to order}, call the Sumitomo Chemical Customer Service Hotline at +1(248)284-4797, or contact us thru our website at WWW.ESPOLEX.COM

References

- ◆ “RJG Technician Training & Master Molder Certification Program – Master Molder 1” binder
- ◆ “Plastics Technician’s Toolbox: Injection Molding – Processing & Troubleshooting”; SPE; 2002
- ◆ “The First Snap-Fit Handbook”; Paul R. Bonenberger; 2000