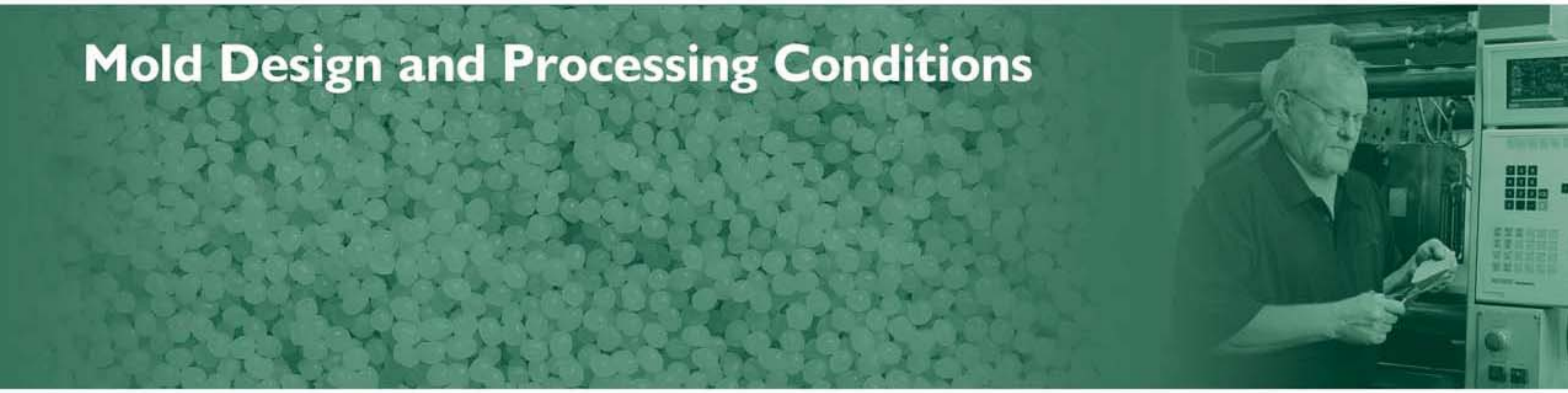
THERMOPLASTIC ELASTOMER

Mold Design and Processing Conditions



A Guide to Processing
Injection Molding Grade Specialty Compounds
English/Standard and SI Metric

Thermoplastic Elastomer

CONTENTS:

- Advantages
- Overmolding Design Considerations
- General Design Considerations
- Processing Conditions
- Mold Design
- Venting
- Cooling

TPE Bondability To Rigid Substrates

Overmolding Material	ABS	PC	PC/ABS	PBT	PBT/PC	PET/PC	PC/PMMA	RTPU	LDPE	PP	PP Long Fiber	PA 6	PS
RTP 2740 S Series										■	■		
RTP 2700 S Series										■	■		
RTP 2800 B Series									■	■	■		
RTP 2800 D Series										■	■		
RTP 6042 Series	■	■	■	■	■	■	■	■					
RTP 6003 Series	■	■	■	■	■	■	■	■					
RTP 6004 Series												■	
RTP 6035 Series										■	■		■

■ Indicates combinations exhibiting very good peel strength (15 pli or greater)

- For simplicity, this chart does not incorporate TPE hardnesses. Typical hardnesses range from 45-75 shore A, but may be customized to each application.

Advantages

Thermoplastic Elastomer Specialty Compounds

RTP Series	Advantages	Disadvantages	Hardness Range	Typical Applications
RTP 1200 TPUR	Toughness/abrasion resistance, tensile/tear properties, high MVTR possible	Low heat resistance moisture sensitivity, process ability	70A – 75D	Adhesives, films,magnetic tape, caster wheel treads, cable jacketing, auto fascia/side cladding, tool handles
RTP 1500 TEEE	Flex fatigue, tensile/tear properties, high temperature performance, MVTR possible, good spring like properties	Moisture sensitivity, price, high hardness range	85A – 80D	Automotive boots and grommets, sporting goods, wire & cable, hose & tube, breath-able films
RTP 2700 S TES	Versatility, wide hardness range, process ability, colorability, low taste and odor, tear strength	Temperature resitance, oil/chemical resistance	5A – 80A	Caster treads, soft touch grips, Pads & cushions, gaskets/seals, Elastic films and straps
RTP 2800 TPV	High temp. compression set oil/chemical resistance broad temperature range, stress relaxation	Tensile/tear strength, abrasion, color ability, low resilience	35A – 50D	Automotive boots, window seals, weather seals & gaskets, grips Co-molded consumer products
RTP 2900 PEBA	Flexability, toughness, good abrasion resistance to low temperature properties low elastomer memories	• Limited Adhesion to various substrates Moisture sensitivity • Higher cost vs. other TPE's	75A – 70D	High performance tubing, athletic shoes and equipment, sporting goods
RTP 6000 Specialty TPE	Advantages will vary based on function specific design. Call your RTP Company engineer for specifics.	Disadvantages will vary based on function specific design. Call your RTP Co. engineer for specifics.	35A – 75A	• Overmold/bonding applications • Wet-grip applications
RTP 2300 E	Outstanding combination of modulus and impact resistance. Excellent sustainability for overmolding applications.	HDT/level of 200 ˚F / 95 ˚C	N/A	• Sporting goods • Tools • Medical

Important Information: The list above is not the complete RTP Company product offering.

Thermoplastic Elastomer (TPE) Advantages:

- Highly compressible in molten state
- Can be used with significant undercuts
- TPE materials respond very nicely with 2nd stage pressure applied in molding process
- Broad process window for basic molding parameters
- TPE materials respond very nicely to higher shear rate in molding
    - Melt viscosity
    - First stage pressure
- Small gates can be used to manage flow into wall sections

Successful Elastomer Molding:

The required adhesion is dependent on the end-use application and environment. Not all overmold grades bond to all substrates. The substrate type determines which RTP Company TPE overmolding grade must be used to achieve the best bond. The end-use environment also influences which grade to select. The expected service life of the product should be considered when establishing adhesion requirements. Initial material selection is the first step.

Shear and peel forces that the product encounters should be considered. Mechanical interlocks and use of undercuts and ribs will create a higher surface area and improve product performance.

- Proper material selection is key to good bonding.
- Do not use only melt temperature to arrive at the optimum set of molding process conditions.
- Avoid condensation of mold steel surface (caused by too low water temperature)
- Shear rate is very important to optimum TPE overmolding.
- Do not oversize the gate(s) – start “steel safe” and slowly increase gate size as required.

Material Flow Behavior

TPE compounds have relatively low viscosity. They are shear responsive and their viscosity is reduced when they are processed at high shear rates. This helps them flow into and fill thin walled sections commonly encountered in TPE overmolding.

General Concepts to Part Design

The wall thickness of the substrate and overmold should be as uniform as possible to obtain cycle times. Wall thickness in the range from 0.028" to 0.150" (0.711 mm to 3.81 mm) will ensure good bonding in most overmolding applications. If the part requires the use of thick sections, they should be cored out to minimize shrinkage problems and to reduce the part weight and cycle time.

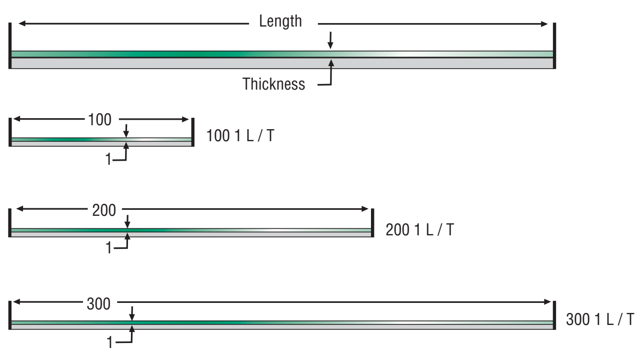
Transitions between wall thicknesses should be gradual to reduce flow problems such as back fills and gas traps. The use of radii (0.020" /0.508 mm minimum) in sharp corners helps reduce localized stress. Deep blind pockets or ribs that are difficult to vent should be avoided. Long draws should have a 3-5° draft to help ejection. Properly designed undercuts however are possible with TPE compounds if the part does not have sharp corners and the elastomer is allowed to deflect during ejection.

For Successful Overmolding:

- Proper gate size/location
- Vent location(s)
- Maximum flow length (L/T)
- Proper surface texture
- Applicable de-gating/part ejection
- Proper shut off technique(s)
- Use mold filling analysis CAE for appropriate complex geometries

Overmolding Considerations

- Use 0.150” (3.81 mm) as a guideline for nominal maximum wall thickness.
- Use of mechanical interlocks on thin wall applications.
- Use 0.028 - 0.032” (0.711-0.812 mm) as a guideline for absolute minimum wall thickness.
- Do not vary TPE component wall thickness by > 4:1.
- Length/thickness >150:1. Thin walls can pose flow challenges and the ability to pack out the component. The flow length to thickness ratio should be 150 maximum. If higher than 150, consider utilizing multiple gates to achieve optimum fill/minimal knit line issues.



Thermoplastic Elastomer Molding Guidelines

- **Press Tonnage:** 2.0 to 3.0 tons (1.8 MT-2.7 MT) of clamp per square inch projected total surface area
- **Barrel Size:** Individual shot size to be 40% - 80% of molding machine barrel capacity
- **Max. Residence Time:** Maximum of 4 to 5 minutes (in the barrel)
- **Screw Selection:** General purpose screws are acceptable
- **Nozzle Diameter:** Must be slightly smaller than the sprue base diameter in mold runner system being run

Overmolding Design Considerations

Thermoplastic Elastomer Specialty Compounds

Vents

- The purpose of venting is to evacuate all entrapped air in the runner system and part cavities.
- Adequate venting is most critical in complex and long length runner systems. Full peripheral venting is recommended - if not possible, vent at the end of fill directly opposite the gate.
- Vent Depth: 0.0005” to 0.001” (0.0127-0.0254 mm) deep - 0.00075” (0.1905 mm) optimum.
- Land Length: Minimum 0.040” (1.016 mm) where vent depth is cut.
- Depth of land (vent run out): > 0.060” (1.524 mm)

Hot Runner Systems

The hot runner design should be balanced for even flow throughout the runner system. Externally heated manifolds work the best for RTP TPE materials. Internally heated systems, which use cartridge heaters, are not recommended as they generally have hot spots which could lead to material degradation. All flow channels should be highly polished with radius corners to minimize the possibility of material stagnation. To maintain high shear, low residence times, and provide for self-clearing, the flow channels should have be .250” to .375” (6.35 mm to 9.52 mm) diameter. Balance volume per shot size.

Hot Sprue Usage For Thermoplastic Elastomer Materials

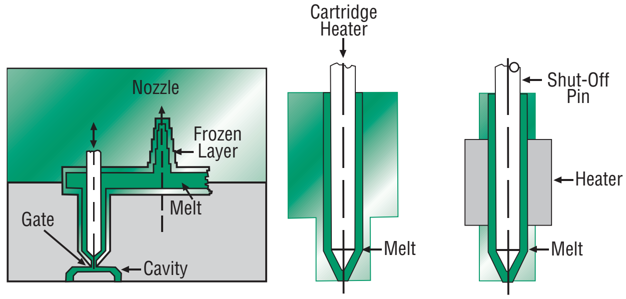
- Can be utilized to reduce scrap in large runner system
- Off-shelf hot sprue commercially available from many suppliers
- Adds no real complexity to runner system design

Benefits:

- Improved economics – less scrap
- Faster cycle time – sprue never freezes off
- 20% to 25% lower total cycle time common in >16 cavity systems
- Can be very significant in large (16 or 32 cavity) cold runner systems
- More repeatable pressure to feed runner system
- Lowers part size variation in critical dimension components

Disadvantages:

- Higher capital cost
- Typically more time to tune mold in on initial start up
- Must be purged at end of molding shift
- Potential for material degradation (if TPE is over heated)
- Typically higher maintenance cost



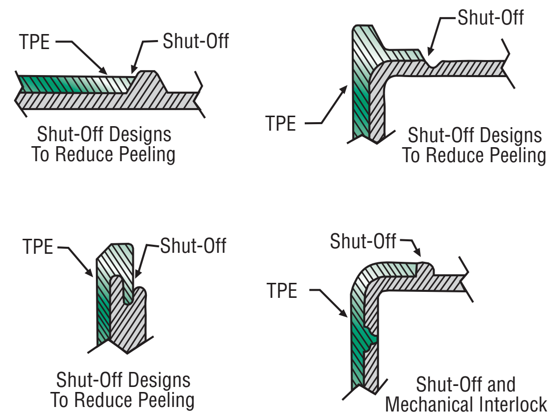
Hot Runner System Hot Sprue Valve Gate

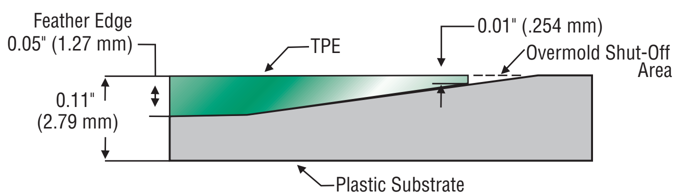
Shut-Offs

Proper shut off design is critical to stop the TPE flow in precise geometry location and prevent undesirable random edge flash

Approach for Good Functionality:

- Design shut off to minimize potential for edge peeling of TPE
- General geometry needs to be a very sharp transition area between molded TPE edge and supporting substrate.
- Resulting TPE geometry must be designed to vent the cavity properly





Overmolding Design Considerations

Thermoplastic Elastomer Specialty Compounds

Shut-Off Recommendations:

- Provide interference fit of 0.002” to 0.004” (0.050 mm to 0.101 mm) - somewhat dependant on cosmetic needs
- Specific plastic substrate ductility
- Heat treat shut off steel to minimum 54 Rockwell hardness
- Consider substrate edge design to “hide” TPE edge from consumer.
- Pre-dry hygroscopic substrate and TPE pellets to avoid porous surfaces nearest substrate interface
- Where appropriate, have the actual shut-offs employed as inserts (helps downstream injection mold maintenance)

Shut-Offs to Avoid

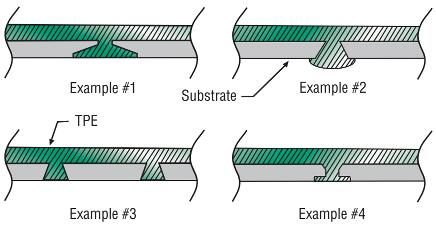
- Avoid using rounded/radiused shut offs
- Component designs where TPE geometry lies high above the substrate (e.g., cliff wall)
- Placing shut offs directly in the mold base
- Building the tool without developing a clear shut off strategy first
- Placing the vent directly at shut off edges – can actually encourage flashing

Fundamental “lock” of TPE plastic compound substrate is to be provided by a combination of three basic methods:

- *Chemical adhesion*
- *Mechanical design techniques*
- *Interlocks*

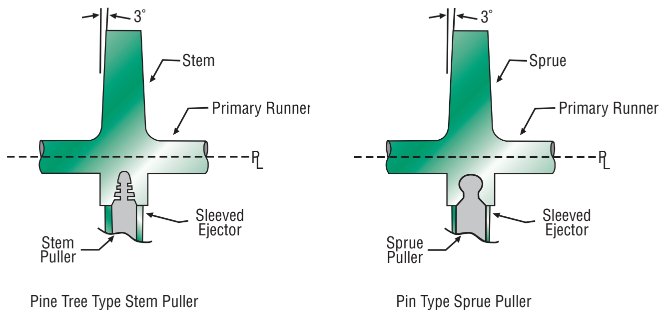
Mechanical Interlocks Offer Major User Benefits:

- Mechanical combination of TPE/substrate geometry (very difficult to separate in use)
- Method to greatly improve component abrasion resistance
- Can be used across a substrate geometry - multiple locations



Sprues and Gate Considerations:

TYPICAL TPE HARDNESS RANGE	SPRUE PULLER TYPE
>60 Shore A	Z-type, Conventional
60 to 90 Shore A	Undercut, Pine Tree

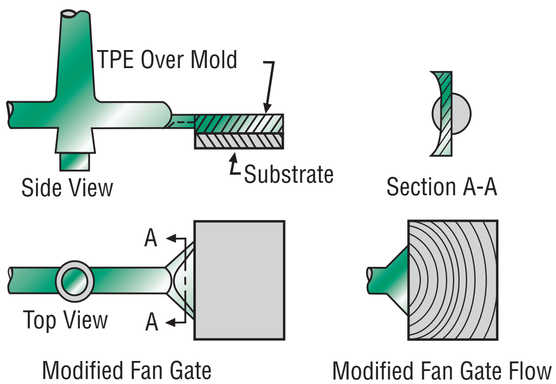


Gating Recommendations:

- Gate at thickest TPE wall cross section
- Optimize gate type for part geometry and cosmetic component preferences
- Gate orientation to component wall should be customized
- Gates can be used in combination with added “flow” channels to insure minimal molded in stress
- A perpendicular gate (at 90° angle) to the melt flow – can cause “jetting” surface effect
- Need to consider exact gate location with respect to knit lines
- Oversized TPE gate(s) – will reduce shear rate in molten TPE flow. Utilize “steel safe” practices by starting small followed by gradual increases

Gating to Avoid:

- Very long flow paths (>200:1 L/T) should be avoided
- Avoid gating into thin cross-section of components
- Typically should not gate near core pins to minimize melt flow obstruction



TPE Ejection and Surface Textures:

- Select ejection mechanism(s) to minimize mechanical damage to surface.
- Consider using pneumatic air poppets as a means to create initial molded TPE surface release from tool steel – excellent method for very thin wall components. Use pneumatic poppets (air) release approach where feasible.
- Determine the most cosmetically important requirements from end user to prioritize surface needs which will guide tool details.
- Use stripper plate to spread stress across molded TPE surface.
- Where feasible, have stripper plate push against the exposed thermoplastic surfaces which are slightly less prone to surface damage.
- Avoid using “sharp” (small diameter) ejector pins in component ejection system.
- Slow ejector speed to minimize the “punch through effect” into the newly molded TPE surface.
- Use texture on mold surface to promote ejection/release.
- If clarity is required, use SPI/SPE #2 finish and use air poppets and a stripper sleeve for ease of ejection.

Surface Texture:

Basic Feel of a soft over-molded surface is affected by three primary factors:

- RTP Company TPE product selection (modulus vs. hardness)
- TPE component wall thickness coupled with modulus
- Structural geometry of plastic substrate under the TPE

The Benefits of Using Custom TPE Surface Textures:

- Can provide a softer feel when using a harder RTP TPE grade
- Can decrease total cycle time (vs. using thicker TPE across section)
- Help reduce scuff and mar
- Can assist in shedding fluid from TPE surface
- Can help “hide” gate marks and cosmetic surface defects
- Can actually increase total surface area that consumer uses with finished product
- Select appropriate RTP TPE to maximize bond strength to selected RTP thermoplastic compound

Machine Type

TPE resins can be molded on reciprocating screw-type injection molding machines. A reciprocating screw machine is preferred as it produces a more homogeneous material and a more uniform melt temperature. It also permits processing at lower temperatures.

Shot Size and Machine Capacity

When running on reciprocating screw-type machines, utilization of 40% – 80% of the barrel capacity is desirable. Although shot weights smaller than 40% have been molded successfully, the material can degrade when the shot weight is too small and excessive heat builds up in the melt.

Screw Type

Following are important considerations in choosing a screw for injection molding TPE resins:

- A general-purpose screw with a length-to-diameter ratio (L/D) of at least 20:1 is satisfactory.

- A compression ratio of 2.0:1 to 3.0:1 is preferred. A 2.5:1 compression ratio is applicable for most situations.

- Rapid-transition (nylon-type) screws are not recommended because of the excessive melt temperature and consequent degradation ofthe resins that can occur with them.

- Exception is RTP 2900 series which with certain grades will process generally better with a nylon-type design screw.

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 entrapments.

- 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 unvented 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 condi-tions. Generally, TPE compounds will flow much further in thinner walls. The one exception to this is the RTP 1200 series (TPE)The flow to thickness ratio should be 150 maximum. Longer flow lengths will require additional gates to keep the L/T below 150.

Ejector Mechanisms

Whenever possible, use air poppets or plate ejection to avoid marking or deforming the TPE materials. Large diameter pins, stripper plates or rings, knock-out sleeves, or core retraction and/or ejection are suitable. All should act on the thickest sections of the part. The location of the ejection system should provide uniform stripping of the part. Avoid using small ejector pins that will deform or push through the part. A cold slug well, opposite the nozzle, should contain a means for pulling the sprue free from the nozzle tip and the sprue bushing.

Sprue and Sprue Puller Design

The sprue should have sufficient draft, from 1º to 3º to minimize drag and sprue sticking. Longer sprues may require more taper (3° - 5°). Typically, the sprue diameter should be slightly larger than the nozzle diameter. Undercut or Z type sprue pullers are satisfactory for all grades. The undercut type is typically used for harder resins (higher than 90A) and the Z-type sprue pullers are more commonly used for softer resins. Permanent surface lubricant treatments have also been used successfully.

Non-Return Check Valves

Non-return check valves prevent the molten polymer in the holding space in front of the screw from flowing back into the screw during the injection cycle. When processing TPE resins, use a free-flowing, sliding check-ring style non-return check valve made of fully hardened H-13 steel, preferably nitrided to retard wear. A fully channeled tip will minimize flow restrictions as TPE resins, like most thermoplastics, the material will degrade when subjected to excess shear at flow restrictions.

Nozzle Types and Tips

Most standard steel nozzle types used with other thermoplastics are satisfactory for molding TPE resins. A straight-through nozzle or a replaceable-tip nozzle with a reverse taper is recommended.

The nozzle should be as short as possible. If a long nozzle is necessary, it should have a large internal diameter proportional to its length. It is essential that the nozzle and sprue bushing mate properly. The nozzle orifice should be slightly smaller (about 20%) than the sprue bushing orifice.

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 TPE compounds 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. Venting in areas below the parting line can be accomplished by allowing the ejector pin to be 0.001” loose on each side. Venting of ribs or pockets can be achieved by venting down an ejector pin. Ejector pin vents are self-cleaning, but should still be maintained and cleaned regularly to remove buildup.

Conventional Runner Configuration and Design

A balanced H-pattern 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 over-packed and flashing may occur. As a result of over packing, parts may also develop high molded-in stresses, which lead to warpage.

System Type	Advantages	Disadvantages
Cold Runner	• Lower tool cost • Easily modified • Enables use of robotics	• Typically governs cycle time • Potential for cold slugs • Potential for sprue sticking • Scrap (though regrindable)
Hot Sprue or Extended Nozzle	• Faster cycle • Minimizes scrap • Easily maintained	• Higher tool cost • Potential material degradation
Hot Runner	• No runner scrap • Faster cycle time • Precise temperature • Control	• Highest tool cost • Purging • Material degradation • Maintenance

Processing Conditions

Thermoplastic Elastomer Specialty Compounds

RTP 1200 Series Thermoplastic Elastomer (TPUR)

Typical Injection Molding Conditions	English/Standard		SI Metric	
Temperatures				
Rear zone	320 - 360	°F	160 - 182	°C
Center zone	350 - 390	°F	177 - 199	°C
Front zone	370 - 410	°F	188 - 210	°C
Melt	365 - 425	°F	185 - 218	°C
Mold	100 - 140	°F	38 - 60	°C
Pressures				
Injection	10000 - 15000	psi	69 - 103	MPa
Hold	5000 - 10000	psi	34 - 69	MPa
Back	50 - 100	psi	0.34 - 0.69	MPa
Speeds				
Fill	0.5 - 1	in/sec	13 - 25	mm/sec
Screw	60 - 90	rpm	60 - 90	rpm
Drying				
Time & Temperature	6 Hrs @ 225	°F	6 Hrs @ 107	°C
Dew Point	0	°F	-18	°C
Moisture Content	0.01	%	0.01	%

RTP 1500 Series Thermoplastic Elastomer (TEEE)

Typical Injection Molding Conditions	English/Standard		SI Metric	
Temperatures				
Rear zone	380 - 410	°F	193 - 210	°C
Center zone	410 - 440	°F	210 - 227	°C
Front zone	420 - 450	°F	216 - 232	°C
Melt	410 - 460	°F	210 - 238	°C
Mold	70 - 120	°F	21 - 49	°C
Pressures				
Injection	10000 - 15000	psi	69 - 103	MPa
Hold	5000 - 10000	psi	34 - 69	MPa
Back	50 - 100	psi	0.34 - 0.64	MPa
Speeds				
Fill	0.5 - 1	in/sec	13 - 25	mm/sec
Screw	60 - 90	rpm	60 - 90	rpm
Drying				
Time & Temperature	4 Hrs @ 200	°F	2-4 Hrs @ 93	°C
Dew Point	n/a	°F	n/a	°C
Moisture Content	n/a	%	n/a	%

RTP 2700 Series Styrenic Thermoplastic Elastomer (TES)

Typical Injection Molding Conditions	English/Standard		SI Metric	
Temperatures				
Rear zone	340 - 360	°F	171 - 182	°C
Center zone	350 - 370	°F	177 - 188	°C
Front zone	370 - 390	°F	188 - 199	°C
Melt	360 - 450	°F	182 - 232	°C
Mold	60 - 100	°F	16 - 38	°C
Pressures				
Injection	10000 - 15000	psi	69 - 103	MPa
Hold	5000 - 10000	psi	34 - 69	MPa
Back	50 - 100	psi	0.34 - 0.64	MPa
Speeds				
Fill	0.5 - 1	in/sec	13 - 25	mm/sec
Screw	60 - 90	rpm	60 - 90	rpm
Drying				
Time & Temperature	2 Hrs @ 175	°F	2 Hrs @ 79	°C
Dew Point	n/a	°F	n/a	°C
Moisture Content	n/a	%	n/a	%

RTP 2800 Series Thermoplastic Vulcanizate Elastomer (TPV)

Typical Injection Molding Conditions	English/Standard		SI Metric	
Temperatures				
Rear zone	340 - 360	°F	171 - 182	°C
Center zone	350 - 370	°F	177 - 188	°C
Front zone	370 - 390	°F	188 - 199	°C
Melt	360 - 410	°F	182 - 210	°C
Mold	60 - 150	°F	16 - 66	°C
Pressures				
Injection	12000 - 18000	psi	83 - 124	MPa
Hold	5000 - 12000	psi	34 - 83	MPa
Back	50 - 100	psi	0.34 - 0.69	MPa
Speeds				
Fill	0.5 - 1	in/sec	13 - 25	mm/sec
Screw	60 - 90	rpm	60 - 90	rpm
Drying				
Time & Temperature	2 Hrs @ 175	°F	2 Hrs @ 79	°C
Dew Point	0.0	°F	-18	°C
Moisture Content	0.03	%	0.03	%

Processing Conditions

Thermoplastic Elastomer Specialty Compounds

RTP 6001 Series Thermoplastic Elastomer (TPE)

Typical Injection Molding Conditions	English/Standard	SI Metric
Temperatures		
Rear zone	340 - 360 °F	171 - 182 °C
Center zone	350 - 370 °F	177 - 188 °C
Front zone	370 - 390 °F	188 - 199 °C
Melt	360 - 450 °F	182 - 232 °C
Mold	60 - 100 °F	16 - 38 °C
Pressures		
Injection	10000 - 15000 psi	69 - 103 MPa
Hold	5000 - 10000 psi	34 - 69 MPa
Back	50 - 100 psi	0.34 - 0.64 MPa
Speeds		
Fill	0.5 - 1 in/sec	13 - 25 mm/sec
Screw	60 - 90 rpm	60 - 90 rpm
Drying		
Time & Temperature	2 Hrs @ 175 °F	2 Hrs @ 79 °C
Dew Point	n/a °F	n/a °C
Moisture Content	n/a %	n/a %

RTP 6003 Series Thermoplastic Elastomer (TPE)

Typical Injection Molding Conditions	English/Standard	SI Metric
Temperatures		
Rear zone	360 - 410 °F	182 - 210 °C
Center zone	370 - 440 °F	188 - 227 °C
Front zone	380 - 450 °F	193 - 232 °C
Melt	380 - 460 °F	193 - 238 °C
Mold	70 - 120 °F	21 - 49 °C
Pressures		
Injection	10000 - 15000 psi	69 - 103 MPa
Hold	5000 - 10000 psi	34 - 69 MPa
Back	50 - 100 psi	0.34 - 0.69 MPa
Speeds		
Fill	0.5 - 2 in/sec	13 - 51 mm/sec
Screw	60 - 90 rpm	60 - 90 rpm
Drying		
Time & Temperature	2-4 Hrs @ 180 °F	2-4 Hrs @ 82 °C
Dew Point	0 °F	-18 °C
Moisture Content	0.03 %	0.03 %

RTP 6004 Series Thermoplastic Elastomer (TPE)

Typical Injection Molding Conditions	English/Standard	SI Metric
Temperatures		
Rear zone	360 - 410 °F	182 - 210 °C
Center zone	370 - 440 °F	188 - 227 °C
Front zone	380 - 450 °F	193 - 232 °C
Melt	420 - 480 °F	216 - 249 °C
Mold	120 - 150 °F	49 - 66 °C
Pressures		
Injection	12000 - 18000 psi	83 - 124 MPa
Hold	5000 - 10000 psi	34 - 69 MPa
Back	50 - 100 psi	0.34 - 0.69 MPa
Speeds		
Fill	0.5 - 2 in/sec	13 - 51 mm/sec
Screw	60 - 90 rpm	60 - 90 rpm
Drying		
Time & Temperature	2-3 Hrs @ 175 °F	2-3 Hrs @ 79 °C
Dew Point	0 °F	-18 °C
Moisture Content	0.03 %	0.03 %

RTP 6042 Series Thermoplastic Elastomer (TPE)

Typical Injection Molding Conditions	English/Standard	SI Metric
Temperatures		
Rear zone	340 - 360 °F	171 - 182 °C
Center zone	350 - 370 °F	177 - 188 °C
Front zone	370 - 390 °F	188 - 199 °C
Melt	360 - 450 °F	182 - 232 °C
Mold	60 - 90 °F	16 - 32 °C
Pressures		
Injection	10000 - 15000 psi	69 - 103 MPa
Hold	5000 - 10000 psi	34 - 69 MPa
Back	50 - 100 psi	0.34 - 0.64 MPa
Speeds		
Fill	0.5 - 2 in/sec	12 - 50 mm/sec
Screw	60 - 90 rpm	60 - 90 rpm
Drying		
Time & Temperature	2 Hrs @ 175 °F	2 Hrs @ 79 °C
Dew Point	0 °F	-18 °C
Moisture Content	0.01 %	0.01 %

* Processing Conditions for RTP 2900 Series Thermoplastic Elastomer (TPE) available upon request.

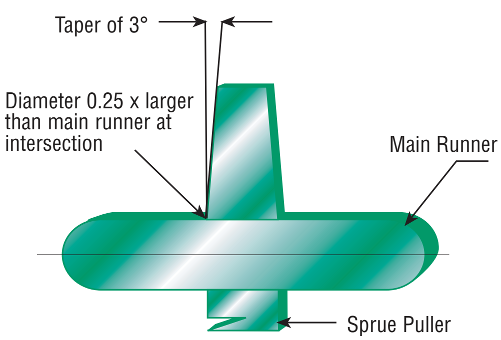
General Mold Design

Injection molds must be properly designed to ensure quality plastic components. Mold design impacts productivity and profitability of your molding operation. This section offers guidelines for designing an efficient injection mold.

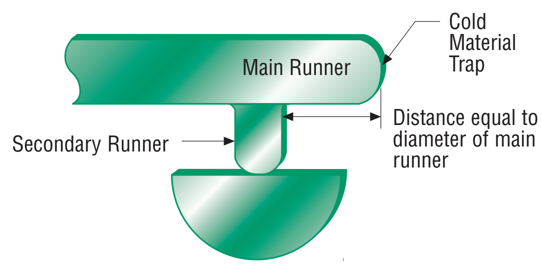
Sprue/Cold Material Trap Design

Sprues connect the nozzle of the injection molding machine to the main runner or cavity. The sprue should be as short as possible to minimize material usage and cycle time. The bushing should have a smooth, tapered internal finish that has been polished in the direction of the draw to ensure clean separation of the sprue and the bushing.

Sprue Puller - An area used to pull sprue out of the bushing as the mold opens.



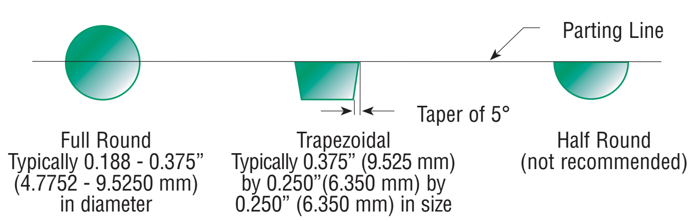
Cold Material Trap - A space provided to trap cooled material during flow.



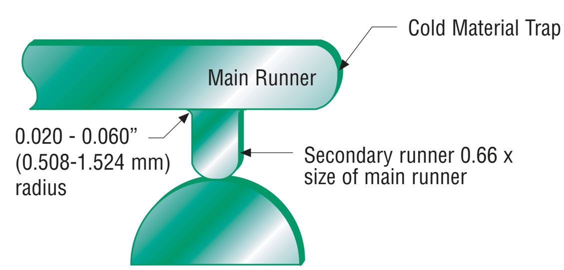
Runner Design

Runner systems convey the melted plastic from the sprue to the gate or part. The most efficient profile for a runner is circular (full-round). A less expensive, yet adequate, profile is a trapezoid, with tapers as shown in the diagram to ensure a good volume-to-surface area ratio. Half rounds are not recommended because of their poor perimeter to area ratio.

Main Runner - A channel that connects the sprue to the gate or part.



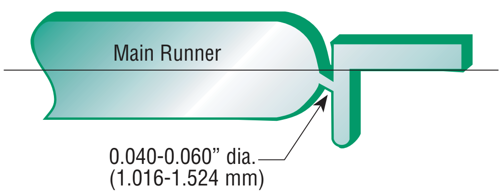
Secondary runner - A runner system located between the main runner and the molded part.



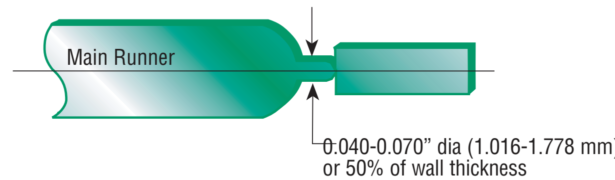
Gate Design

The gate serves as the entrance to the cavity and should be designed to permit the mold to fill easily. A cavity can have more than one gate. Gates should be small enough to ensure easy separation of the runner and the part but large enough to prevent early freeze-off of polymer flow, which can adversely affect the consistency of part dimensions. A variety of gate designs and locations are shown below:

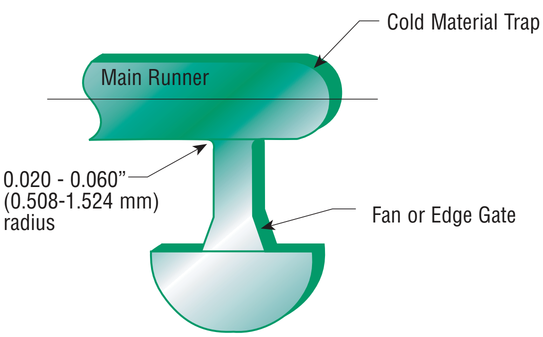
Submarine or Tunnel Gate - An edge gate located below the parting line or molded surface.



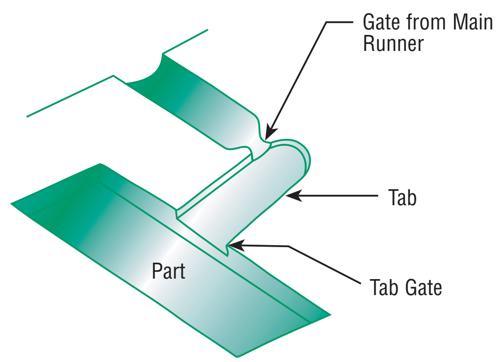
Pinpoint or Restricted Gate - A restricted opening between the runner and molded part. Normally used with thin wall parts.



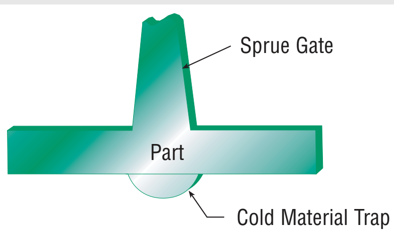
Fan or Edge Gate - A common gate located in the sidewall of the part to prevent restriction of resin flow. Normally used with multi-cavity, two-plate molds.



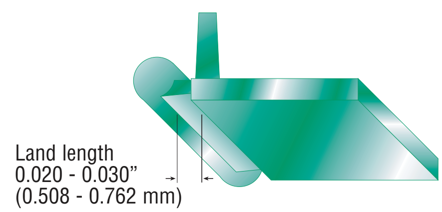
Tab Gate - Used for melt orientation when a large volume is needed for mold fill. The tab helps avoid surface splotches due to high shear, direct gating, or jetting.



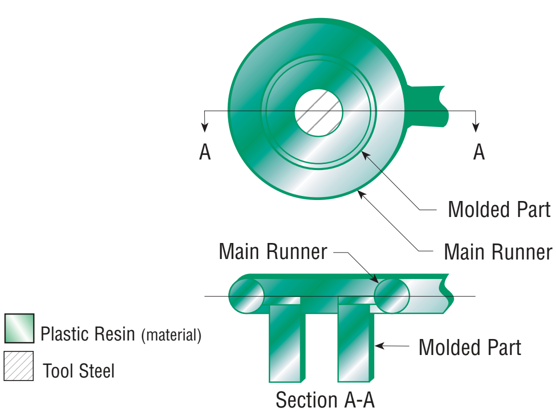
Sprue Gate - Recommended for single cavity molds requiring symmetrical filling. Usually used with circular parts.



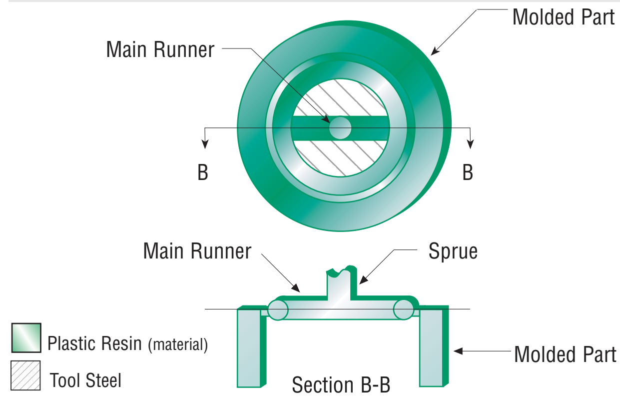
Flash Gate - A long, shallow, rectangular edge gate.



External Ring Gate - A system used when concentricity and a smooth interior surface are important. Can be used in multi-cavity molds.



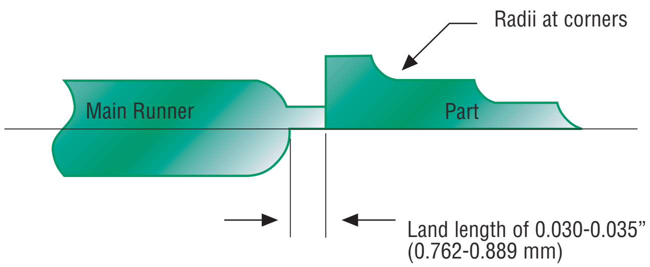
Internal Ring Gate - A system used with large circular parts when concentricity and smooth outer surface are required. Can only be used with single-cavity molds.



General Mold Design

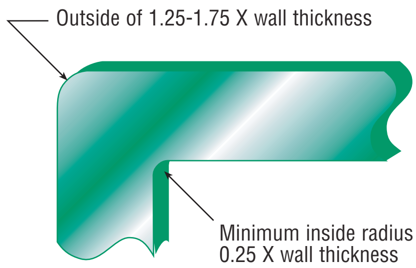
Thick to Thin Wall Gating

Avoid sudden changes in wall thickness by using transition zones to eliminate stress concentrations and reduce sinks, voids, and warping in the molded part.



Adjoining Walls

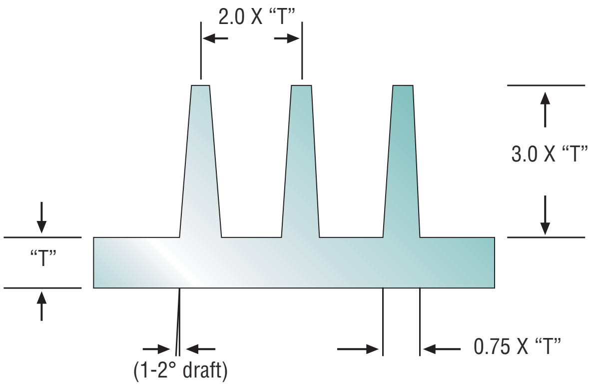
The connection point between two molded surfaces.



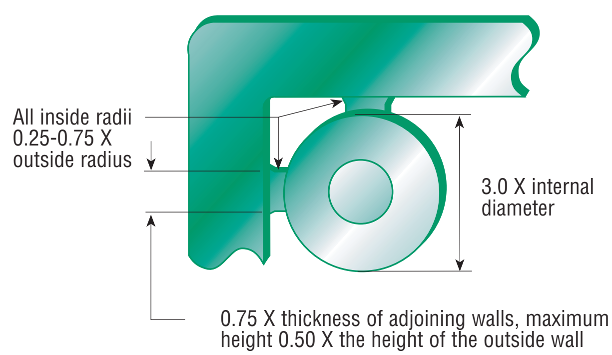
Ribs

Ribs should follow the proportional thickness guidelines shown below. If the rib is too thick in relation to the part wall, you may experience sinks, voids, warpage, weld lines, and longer cycle times. Position ribs in the line of flow to improve filling and prevent air entrapment.

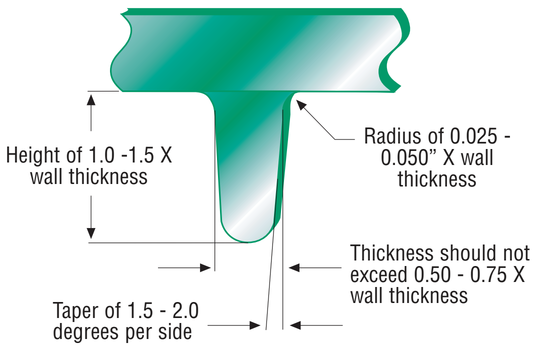
Rib Details



Support Ribs



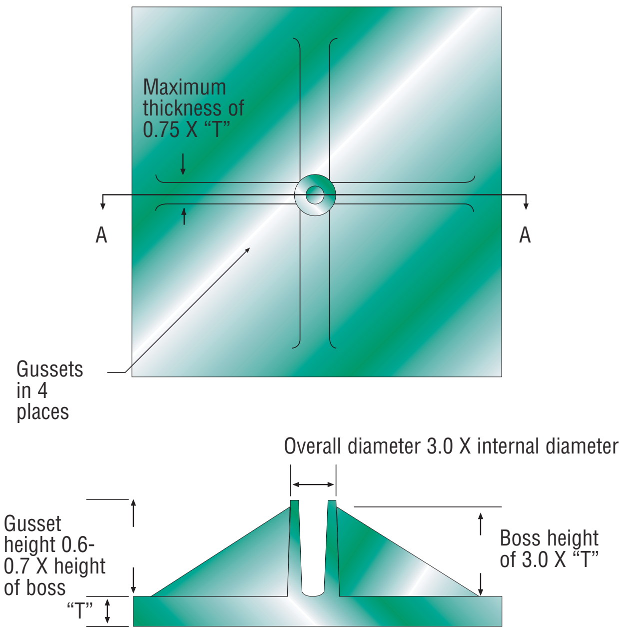
Boss or Rib - A reinforced or protrusion on a mold part for strength or alignment during assembly or fastening.



Bosses & Gussets

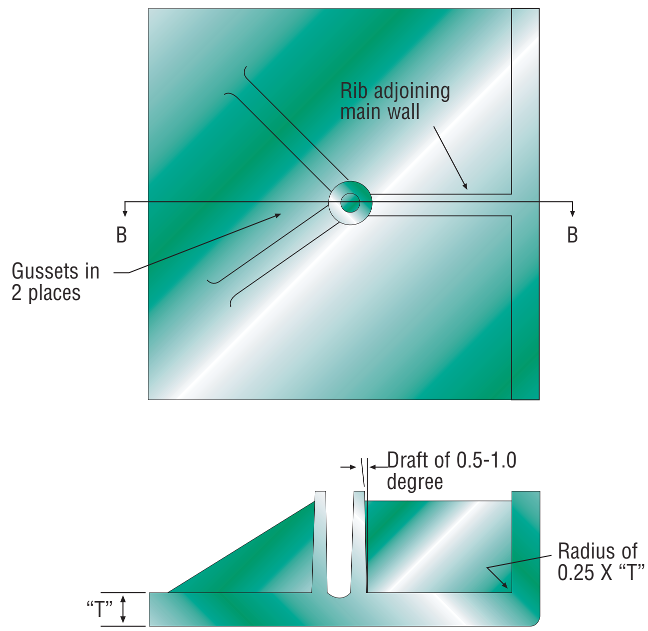
Bosses are used in parts that will be assembled. Connect the boss to a wall or rib with a connecting rib as shown in Figure 1. If the distance of the boss from the wall makes a connecting rib impractical, design the boss with gussets as shown in Figure 2.

Free Standing Boss With Four Gussets



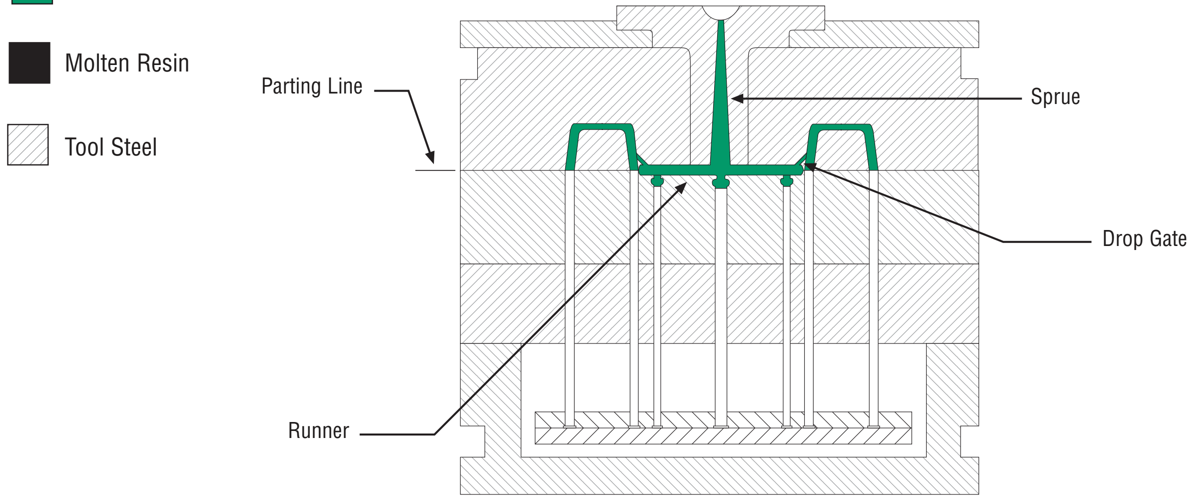
Section A-A

Free Standing Boss With Two Gussets

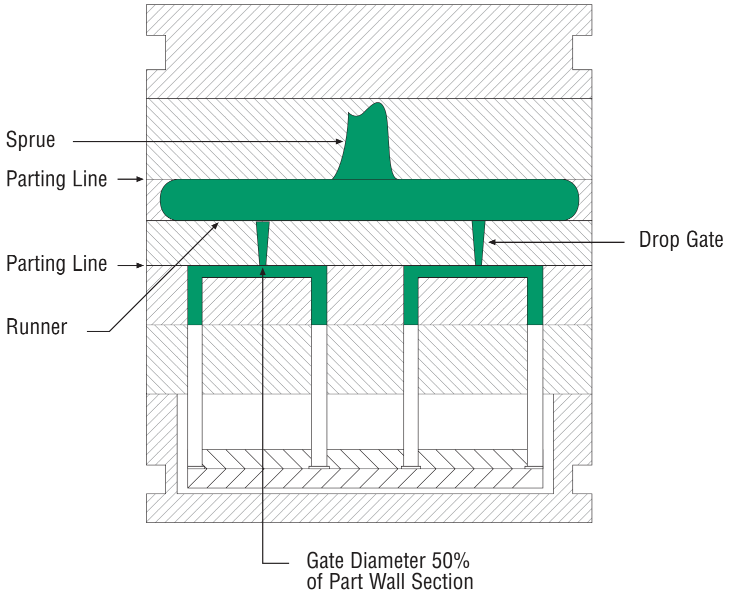


Section B-B

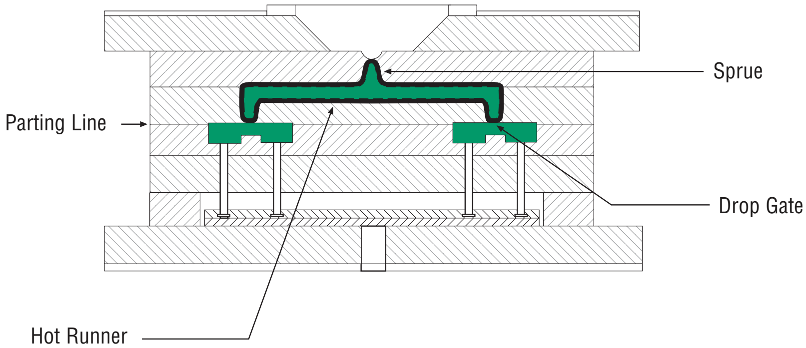
General Mold Design



Two Plate Mold



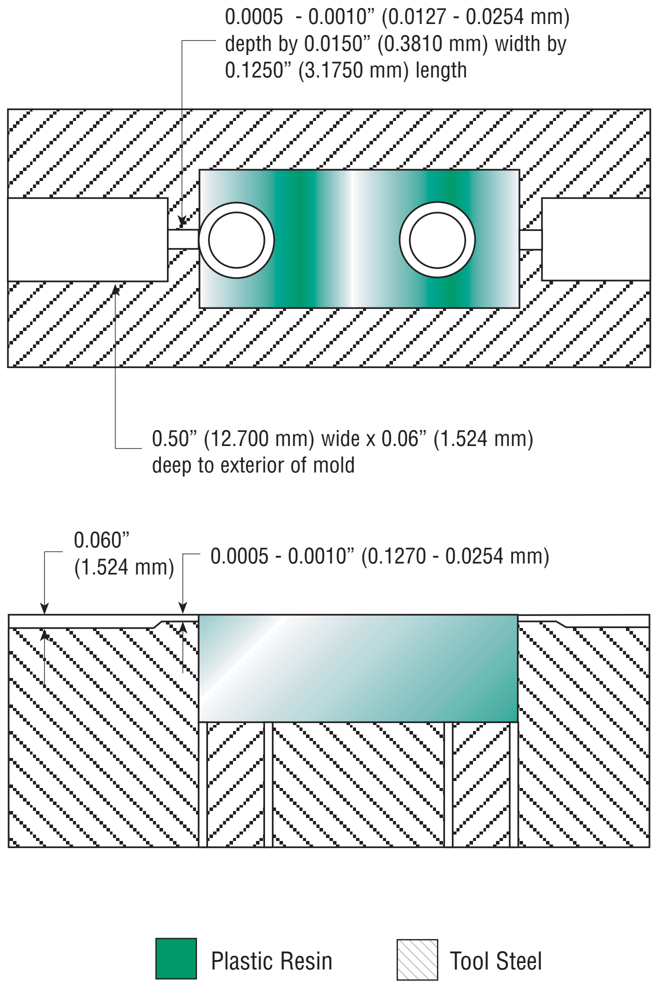
Three Plate Mold



Insulated Runner Mold

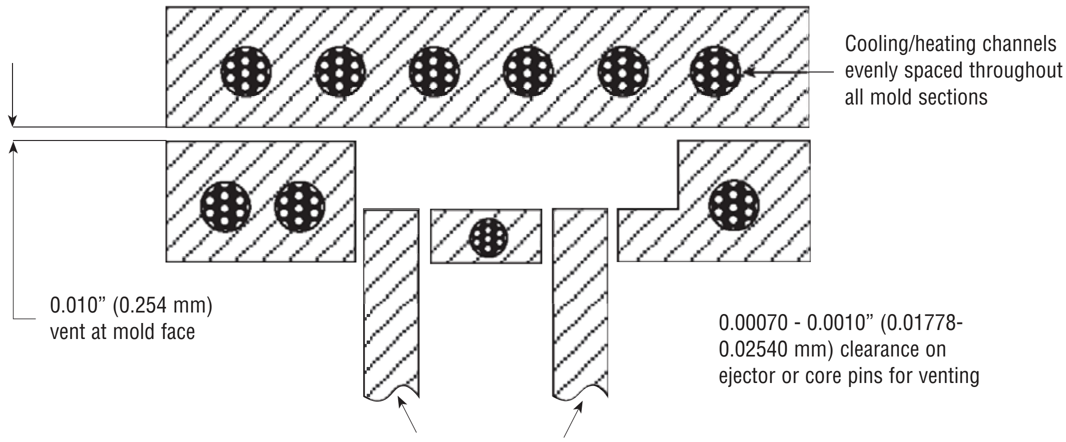
Venting

Proper venting of cavities is very important. Inadequate venting can result in gas burns, poor weld line strength and nonfilled parts. Too much venting can result in excessive flash and poor weld lines due to inadequate pressure buildup. Venting should primarily be located at the last point of fill and where weld lines occur. Vent size depends on the viscosity of the polymer and can vary from 0.0005 - 0.0030" (0.0127 mm to 0.0762 mm) deep. Venting can also be used around knockout pins, moving cores and mold inserts.



Cooling

Molds must be provided with adequate cooling to take advantage of the faster cooling rates of reinforced compounds. Poor cooling results in rising mold temperatures and longer cycle times. Inadequate heating can result in voids, shorts and poor surface finish. Cooling and heating channels should be located directly in the mold inserts and cores if mold design permits.



Ejector Pins: ***Should be located on the heaviest sections of the part to minimize distortion when it leaves the core They should be balanced as much as possible over the part’s surface. Reinforced thermoplastics require more pins due to lower mold shrinkage and greater potential for drag during ejection.***

PROBLEM

SUGGESTED REMEDIES
Perform in numerical order by column

SUGGESTED REMEDIES	Flash	Jetting	Knit Lines	Non-Uniformed Color	Over Mold Breaks Through Substrate	Shorts	Surface Sinks	Surface Splay	Parts Sticking in Cavity	Poor Adhesion	Warp
Add Mechanical Interlocks										5	
Add Surface Texture									5		
Add Pneumatic Air Poppets									8		
Check for Material Contamination				3				2		6	
Check Mold Shut-Off	5										
Check Vent Depth	6		6					5			
Clean Mold Surface			1					4			
Decrease Back Pressure											
Decrease Injection Pressure					2						
Decrease Injection Speed	4	1			1			3			
Decrease Hold Pressure	1								1		5
Decrease Material Temperature	3				3		2		2		
Decrease Mold Temperature (Cavity)									3		
Decrease Shot Size	2				5				4		1
Dry Material	7						7	1		7	
Gate in Thickness Cross Section						6	5				
Incompatible Materials					4					1	
Increase Amount of Gates						7					
Increase Back Pressure				1							
Increase Hold Pressure						4	1				
Increase Injection Pressure							3				
Increase Injection Speed			2			5				2	
Increase Material Temperature		3	4	2		3				4	
Increase Mold Temperature		2	3			2				3	2
Increase Shot Size						1	4				
Increase Substrate Support				4							3
Move Gate Location		4	5	5			6		6		4
Redesign Ejection Mechanism									7		

Imagineering Plastics®

Your Global Compounder Of
Custom Engineered Thermoplastics

Corporate Headquarters:

RTP Company
580 East Front Street
P.O. Box 5439
Winona, MN 55987-0439 USA

Phone:
(800) 433-4787 or
(507) 454-6900

Fax:
(507) 454-4629

Website:
www.rtpcompany.com

E-mail:
rtp@rtpcompany.com

Production Facilities:

Winona, Minnesota

Sauk Rapids, Minnesota

South Boston, Virginia

Fort Worth, Texas

Indianapolis, Indiana

Monterrey, Mexico

Beaune, France

Ladenburg, Germany

Singapore

Suzhou, China