GE Plastics

GE Engineering Thermoplastics Injection Molding Processing Guide

Injection Molding Processing Guide

Introduction

About This Injection Molding Processing Guide

This injection molding guide contains general injection molding parameters that apply to all GE engineering thermoplastic resins.

Mold Design

Processing

Troubleshooting Tips

Available on request are separate product sections with processing information specific to each GE resin family as listed below.

CYCOLAC® resin	CYC-400
CYCOLOY® resin	CYL-425
ENDURAN™ resin	END-200
GELOY® resin	GEG-200
LEXAN® resin	CDC-500
NORYL® resin	CDX-811
NORYL GTX® resin	CDX-200
SUPEC® resin	SUP-300
ULTEM® resin	ULT-210
VALOX® resin	VAL-151
XENOY® resin	X-106
Custom Engineered Products	CEP-200

Call 1-800-845-0600 for all literature requests.

Injection Molding GE Thermoplastics

New dimensions in processing latitude, adaptability to existing equipment, economics and desirable end-product characteristics are presented to the injection molder of GE Plastics resins in this processing guide. Consistent in composition and quality, dependable across a wide range of industrial conditions, GE thermoplastic materials incorporate the flexibility, feasibility and assurance processors required to meet the sophistication and selectivity in marketplaces today.

GE resins have outstanding processing characteristics and many were especially designed for injection molding. However, these resins – like all thermoplastic materials – are not indestructible and must be processed appropriately. This is especially true with GE Plastics flame-retardant resins which have excellent processing characteristics, require proper temperature control within the latitudes specific for each grade. It is important that machinery, processing parameters and molds be utilized under conditions which give sufficient temperature control, minimizing shear heat, material hang-up and resistance to flow.

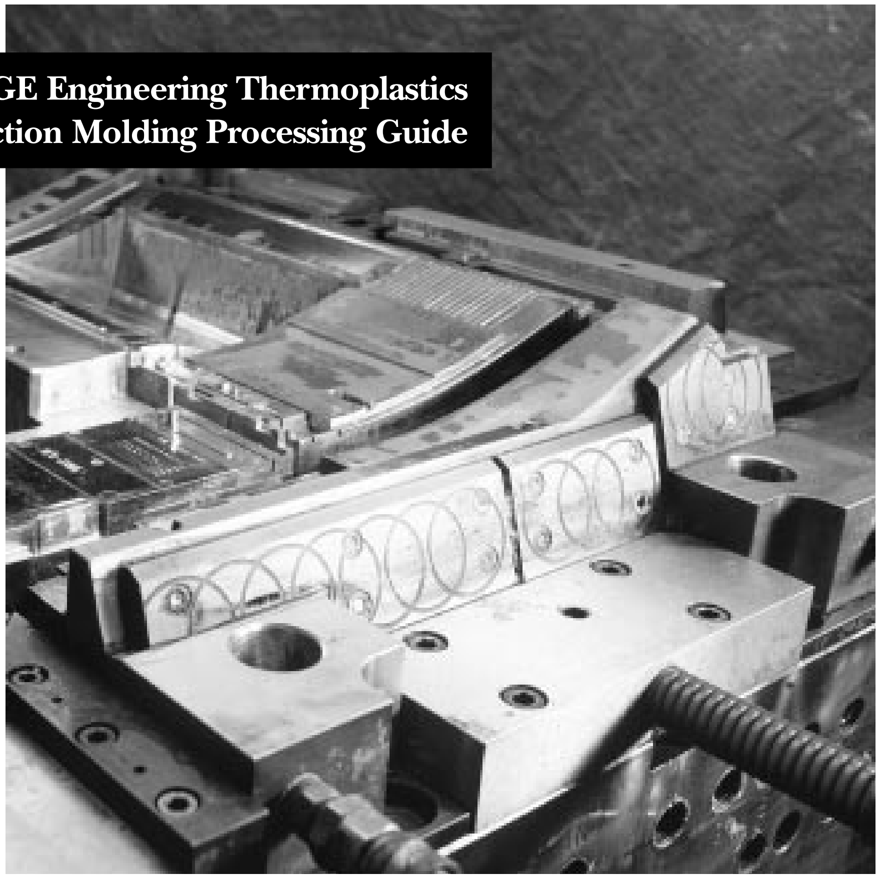
GE Plastics' resins are converted into final parts by a melt process. Generally this is the injection molding process where a plastic melt is injected at high pressures into a precision mold. In addition to this being a high pressure process, it is also a high temperature process. GE Plastics resins can be processed at temperatures ranging from 425˚F (CYCOLAC® resin) to 800˚F (ULTEM® resin). Proper molding practices for GE Plastics resins must be employed to prevent excessive product degradation.

Introduction

This Injection Molding Processing Guide is offered as an informational service to customers of GE engineering plastics and users of products made from these plastics. While GE has used reasonable efforts to provide accurate information in this publication, GE expressly disclaims any liability for damages, costs or losses resulting from reliance on the contents of this Injection Molding Processing Guide. It is the responsibility of the product manufacturer to determine the suitability of any design or product for its intended purpose.

The standards and specifications discussed in this Injection Molding Processing Guide are complex and subject to revision. The general information contained in this publication is an overview only and is not intended to substitute careful and independent examination of applicable standards and specifications. Adequate end-use environmental testing of finished parts must always be conducted.

GE Plastics



GE Engineering Thermoplastics
Injection Molding Processing Guide

Injection Molding
Mold Design

Contents

Mold Materials

Steel selection in tooling can be as critical to the success of a plastics application as the selection of resin is to the end use performance requirements of the molded product. Just as resins are formulated to meet performance requirements in plastics applications, steels are alloyed to meet specific performance requirements in use.

Some applications may require a mold steel with high hardness and wear resistance for parting line durability, while others will require a mold steel with higher toughness for resistance to mechanical fatigue. In general, steels delivering higher hardness and wear resistance properties are those that tend to be more brittle, and in almost all cases, a steel with greater toughness will deliver some reduction in resistance to steel-to-steel wear (adhesive wear) and abrasive resistance to resins containing glass fibers or mineral fillers.

A moldmaker may select a stainless steel to mold a resin that could be aggressive to most other steels. Listed in Table 1-1 on page 1-5 are some of the most commonly used materials in mold building.

Parting line integrity will typically be greater with higher hardness steels (Rockwell 55 or higher), and where steel-to-steel shut-offs produce coring. One or both steel faces should be in the hardness ranges of Rockwell 55 to Rockwell 58.

For abrasion protection from glass or mineral filled resins, it is suggested that gate inserts of A-2, D-2 or M-2 steel be considered with an abrasive-resistant steel be inserted in the mold core opposite the gate.

P-20 Steel

While there is no “general purpose” steel for plastic molds, P-20 steel has been regarded as the workhorse of the industry. Supplied in the pre-hardened state at Rc 30-32, it is very tough, yet fairly easily machined. It is a good steel to consider in applications where cavity sizes exceed 12 × 12 × 12 inches (303.6 × 303.6 × 303.6 mm), since the cost and associated risks of heat treating blocks of this size may be prohibitive. P-20 steel is also chosen in smaller cavity sizes to eliminate the time and expense of heat treatment when it is anticipated that the mold will not exceed 500,000 cycles.

When constructing a mold of P-20 steel where slides, lifters or other cams or moving components are necessary, it is suggested that these moving steel components be made of steels with different alloying and hardness to reduce galling or high adhesive wear. A common practice in large molds of P-20 steel is to employ slides or lifters of H-13 steel that is heat treated to Rc 50-52 or to employ localized wearing surfaces of steels in the Rc 55 through Rc 58 ranges, or both.

H-13 and S-7 Steels

These steels offer an extremely high degree of toughness and mechanical fatigue resistance with a perceived higher toughness in H-13 (Rc 50-52) but better durability in S-7 because of higher hardness (Rc 55-57). Neither exhibits exceptional abrasion resistance from glass or mineral resin fillers. Gate inserts of A-2, D-2 or M-2 are commonly used in filled resin applications.

It is common for H-13 to be chosen in cavities larger than 8 × 8 × 8 inches (202.4 × 202.4 × 202.4 mm) where a higher degree of hardness and toughness over P-20 is required. Smaller cavities and cores are commonly constructed of S-7. S-7 can be heat treated in an air quench in small cross sections of 2 1/2 inches (63.25 mm) or less, and offers very good dimensional stability through this process. Large cross-sections of H-13 and S-7 must typically be quenched in oil.

Corrosion Protection

Nickel plating or stainless steels may be needed to help prevent mold corrosion when molding in a high humidity environment. Corrosion is most likely to occur with a cold mold where condensation, then oxidation may occur, or when using a molding material that may emit a gas that is aggressive to most steels. Nickel plated or stainless steel molds are not normally required to mold GE resins because mold temperatures should be no cooler than 140˚F (60˚C) and only a few injection molding grades of GELOY resin have an aggressive (PVC) component. It is generally suggested that, if there may be occasions of long-term mold storage, where corrosion protection beyond preventative spraying may be necessary, nickel plating may be employed. Electroless nickel plating offers excellent chemical

(Continued on page 1-6)

Table 1-1. Commonly Used Materials in Mold Building

Steel Type	Hardness†	Properties/Typical Application	Drawbacks
M-2	Rc 62-64	Extreme hardness, abrasive and adhesive wear resistance with good toughness. Gate inserts, core pins, shut-off or parting areas.	Difficult and costly to machine and grind due to abrasion resistance.
D-2	Rc 57-59	High hardness, good abrasion resistance. Gate inserts and areas of cavities where high wear from glass or mineral fillers can occur.	Brittle and somewhat difficult to grind, assemble.
A-8	Rc 56-58	Very good adhesive wear resistance, very high toughness. Slides, lifters, cams.	Fair abrasion resistance.
A-6	Rc 56-58	Very good heat treat stability, high hardness, compressive strength. Considered good general purpose air hardening steel.	Moderate ductility.
A-2	Rc 55-57	Good heat treat stability, good abrasion resistance.	Moderate ductility.
S-7	Rc 55-57	Very good mechanical fatigue resistance, excellent hardness/toughness properties.	Moderate adhesive, abrasive wear resistance.
O-1	Rc 56-58	Considered general purpose oil hardening steel. Fair to good adhesive wear resistance. Small inserts and cores.	Medium to low toughness.
L-6	Rc 55-57.	Very good toughness, oil hardening with good heat treat stability.	Medium hardness, medium to low wear resistance.
P-5	Rc 55-57	Highly malleable. Used as hobbing steel.	Case hardened. Very low core hardness, low durability and heat treat stability.
P-6	Rc 55-57	Easily machined, welded.	Low heat treat stability. Medium to low durability.
P-20	Rc 28-32	Considered the standard of the industry. Very tough. Easy to machine. Good for larger cavities.	Moving steel components should be made of steels with different alloying and hardness to prevent galling or high adhesive wear.
H-13	Rc 50-52	Very high toughness.	Low hardness.
SS 420	Rc 48-50	Very good chemical resistance.	Low hardness, little mechanical fatigue properties, low thermal conductivity.
Amco* Metal 945	Rc 31	High thermal conductivity. Used in areas of cavities or cores needing high degrees of cooling stability.	Durability, wear resistance.
Amco* Metal 940	< Rc 20	Very high thermal conductivity when compared to Amco 945.	Very low hardness, durability.
Aluminum QC-7 6061-T651	 Rc 16 Rc 8	Used for tools where volumes are lower. Softer than steel it can be machined faster and at a lower cost.	Does not have the durability of steel.

* Trademark of Crucible Steel
† Rc = Rockwell

protection and is relatively inexpensive when compared to chrome or other techniques, in addition to offering more ease in demolding with most GE resins.

Finally, nickel plating can allow for steel selection offering higher mechanical properties such as toughness, hardness, abrasion or adhesion wear resistance, and higher thermal conductivity than stainless steels.

Prototype Tooling

Soft, lower-cost molds can serve a valuable function by providing pre-production parts for marketing studies, manufacturing assembly requirements, dimensional capabilities or by giving the designer an opportunity to evaluate some unusual function.

All casting and plating processes require a model which will be faithfully reproduced. The quality and durability of prototype tooling depends on the process. Some molds may produce fewer than 100 pieces, whereas others function for many thousands of pieces. The cost and timing of the project may be the deciding factor in which method is used.

Some important molding information can also be gained which can be later translated to the production mold. However, since the thermal and other properties of the prototype mold are often quite different than those of the production mold, the processing parameters and part properties should not be expected to exactly duplicate these in production. Some common forms of producing prototype molds are as follows:

Conventional Machining Practices

- Steel (unhardened)
- Aluminum
- Brass

Casting Process

- Kirksite* – a metal casting material
- Aluminum
- Plastics, epoxies

Liquid Plating Process

Intricate shells can be nickel-plated on a master. These are later backed up and inserted into a mold frame.

Flame Spraying

Flame spray metals can quickly produce a 1/8 inch (3.16 mm) thick shell which is further backed up and placed into a regular frame. A variety of metals which come in wire form can be utilized into the process.

*Kirksite is a Trademark of NL Industries

Mold Filling Pressure

Molding parameters are related to part thickness, part geometry and the chosen gate system. Additional information on mold filling pressures and temperatures can be found in Typical Processing Parameters table included in each individual product family publication. (See page ii)

Computerized Mold Filling Analysis

To obtain optimal flow and balanced gating in conjunction with the lowest pressure drop during filling, a mold filling analysis should be done. There are several types of software commercially available for this purpose.

Sprues and Runners

Cold Sprues

It is suggested that a cold-slug well be provided at the base of the sprue to receive the cold material first emerging from the nozzle. Well diameter should typically be equal to the largest diameter of the sprue, with depth 1-1/2 times this diameter. Wells should also be furnished in the runner system by extending the runner at least 1-1/2 times the runner diameter beyond every 90˚ turn. See Figure 1-6 on page 1-9.

A standard sprue bushing of 1/2 or 3/4 inch (12.7 or 19.05 mm) taper per foot should have a minimum “O” dimension of 7/32 inch (5.56 mm) diameter. The diameter of the sprue at the parting line should be equal to or slightly larger than, the runner diameter. An oversized sprue diameter at the runner intersection may result in a longer molding cycle. The sprue bushing should have a 1/32 to 1/16 inch (0.79 to 1.59 mm) radius at the runner intersection. A reverse-tapered or “dovetailed” cold well will act as a sprue puller at the runner intersection (Figure 1-1).

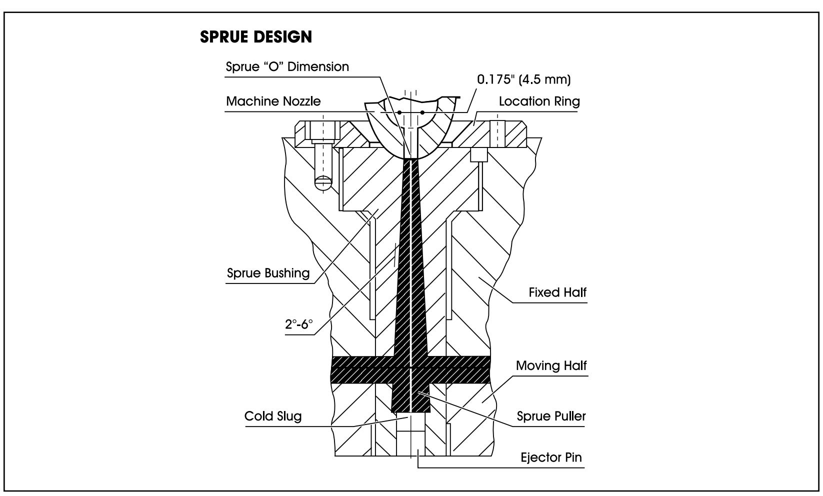
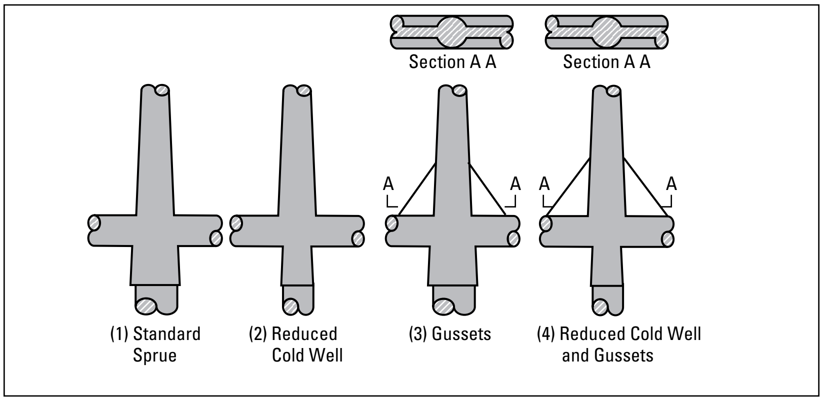


Figure 1-1. Sprue Design.

Cycle time may be reduced by more positive sprue extraction (Figure 1-2), attained by reducing the diameter of the cold well (2) and/or adding gussets (3) or (4) on either side of the sprue.

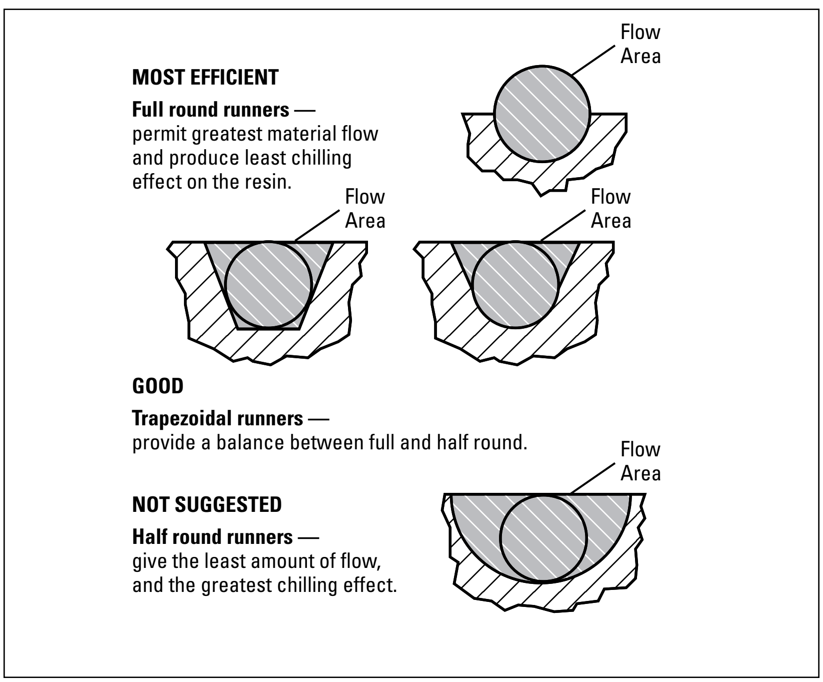
Figure 1-2. Sprue Puller Reverse Taper Modification.



Cold Runners

Full round and trapezoidal runner profiles are suggested for GE Plastics resins (Figure 1-3). The proper choice of the runner diameter depends on the runner length (Figure 1-4). Trapezoidal runner profiles are suggested for three-plate molds (Figure 1-5).

Figure 1-3. Runner Design.



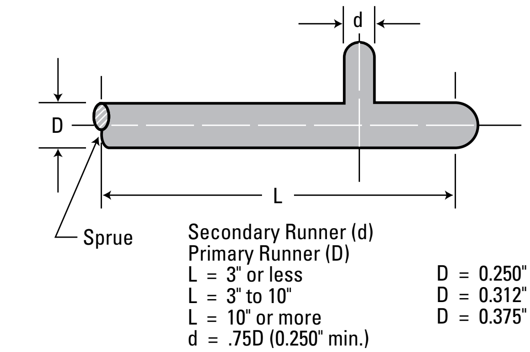
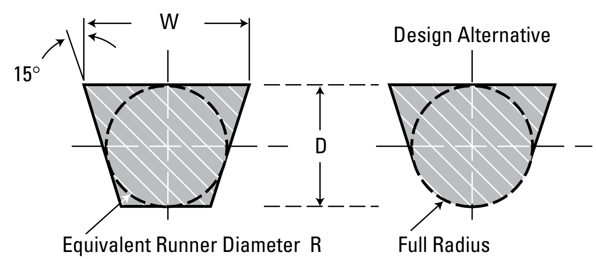


Figure 1-4. Suggested Runner Diameters.



R		W		D	
in.	mm	in.	mm	in.	mm
3/16	4.8	1/4	6.4	3/16	4.8
1/4	6.4	5/16	7.9	1/4	6.4
5/16	7.9	7/16	11.1	5/16	7.9
3/8	9.5	1/2	12.7	3/8	9.5

Figure 1-5. Trapezoidal Runner Profiles.

Figure 1-6 shows cold well size and location of runner vents.

On multiple-cavity molds with primary and secondary runners, the primary runner should extend beyond the intersection of the secondary runner in order to provide a cold well for the runner flow front.

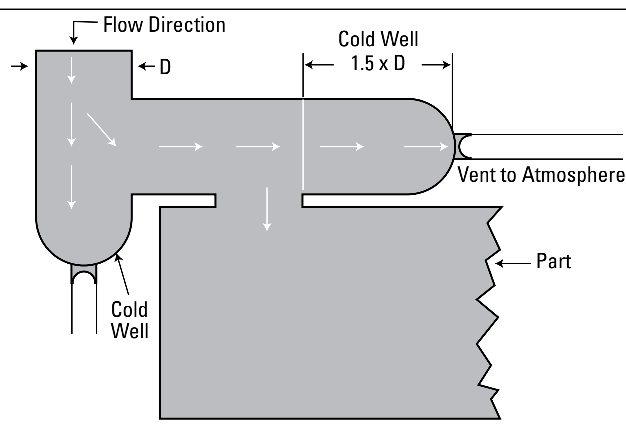
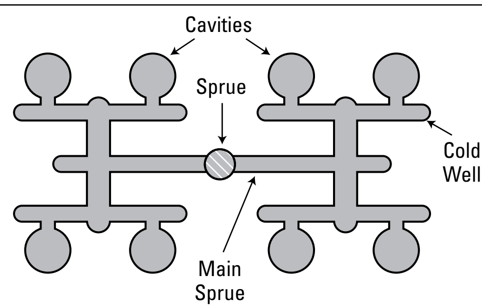


Figure 1-6. Mold Design Runners.

Runner length should be kept at a minimum. Parts requiring close dimensional control in multi-cavity molds should have balanced runner systems (Figure 1-7). Close tolerance parts should not be designed into family mold layouts.

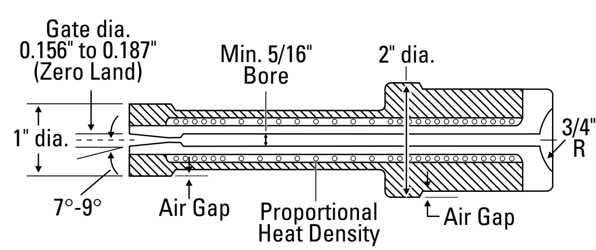
Figure 1-7. Eight-Cavity Balanced Runner.



Hot Sprues

GE resins can be processed in most types of heated sprue bushings. However, it is suggested that externally heated bushings be used (Figure 1-8). Heater power of 50 watts/cubic inch of heated steel is suggested for hot sprue bushings and manifolds. In addition, contact areas should be stress-relieved and, where possible, should utilize stainless steel or titanium inserts for insulation. Precise temperature control at the tip of the drop must be maintained to help ensure proper molding. Heaters should be located along the entire length of the drop and placed in such a way as to provide the proper amount of heat directly to the gate area. The thermocouple should be located as near the tip as possible (Figure 1-9). The amount of surface contact between the drop and the mold should be kept to a minimum (Figure 1-8). This can be accomplished by relieving most of the steel around the drop. The inside diameter of the drop should be no less than 5/16 inch (7.94 mm).

Figure 1-8. Heated Sprue Bushing.



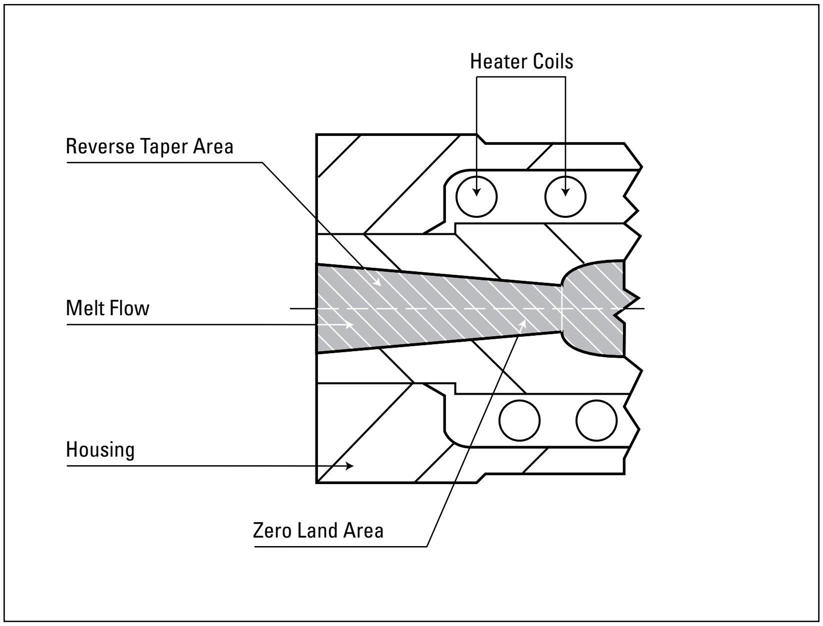


Figure 1-9. Enlarged View of Hot Sprue Bushing Tip.

Hot Runners

The following information is intended only to offer general guidance in the selection and design of hot runner systems, since specific configurations will vary according to individual application requirements.

Hot runner systems have been used successfully for small parts in multi-cavity molds and for large multi-gated parts such as business machine housings and automotive instrument panels.

In the hot runner system, the resin is injected from the machine nozzle into a heated distribution manifold. The resin is kept in the molten state as it flows through the internal passages of the manifold to the bushings and into the mold cavities. The manifold is an extension of the heating cylinder; therefore, it is important that precise temperature control be provided. Internally heated runnerless molding systems are not suggested because of the no-flow areas and higher pressure drops inherent in these systems. This could lead to extended residence time, material degradation and excessive pressures or non-fill conditions.

Hot Runner Benefits

- Although tooling can be more costly, it offers the opportunity for thinner, more economical wall sections based on design rather than flow constraints.
- The part can be gated in several locations enabling design flexibility. Hot runner systems can effectively shorten flow lengths in long part or multi-cavity tools.
- Potential for eliminating regrind and secondary operations (no runners and sprues to remove and regrind).
- Reduce cycle times by eliminating sprue/runner intersections and the need to wait until sprue sets up.
- Hot melt at cavity helps to improve quality. No significant temperature and pressure losses in runners. Fewer molded-in stresses.

Individually-controlled heat zones are strongly suggested. Considerations for hot runner systems are listed below:

Hot Runner Systems

1. Passages should be externally heated and designed to eliminate hot spots (i.e., 4 heaters symmetrically positioned around flow passages).
2. To attain proper manifold heating, a minimum of 50 watts/cubic inch of steel should be applied with heaters distributed evenly throughout the manifold.
3. Manifold passage should be 0.50 inch (12.7 mm) minimum. Large parts and long flow lengths require larger diameters.
4. Passages should be streamlined with no dead spots for material hang-up and degradation. Where there is a corner, the manifold flow channel should have contoured end plugs.
5. Voltage proportional temperature controls are suggested to maintain uniform temperatures.
6. Nozzles should be short and straight through. Nozzles over 6 inches long should have 2 zones of temperature control.
7. The manifold block and drops should be properly insulated from mating steel with 0.03 inch (0.79 mm) air gap with minimum metal-to-metal contact and stainless steel or titanium support pads.
8. Nozzle drops should have external heaters that extend at least even with or beyond the minimum nozzle orifice point.
9. There should be separate closed loop controllers for each nozzle.
10. Resin melt temperature in the manifold and the drops should be the same as the melt temperature in the cylinder.
11. Insulated runner systems are not appropriate for use with engineering thermoplastic resins.
12. With ULTEM resins beware of particular valve gated hot runner systems due to inherent hang-up areas that may cause part streaking.

Gating

The basic considerations in gate location are part design, flow, and end-use requirements. As a general guideline, the following points should be kept in mind:

- Large parts that require multiple gates should include gates that are positioned close enough together to reduce pressure loss. This will help provide good weld line strength by minimizing cooling where the leading edges of resin flow come together. Gating dimensions should provide resin fill at reasonable pressures and fill speeds.
- Gate land lengths should be kept as short as possible.
- An impinging gate will help ensure that the incoming flow is directed against the cavity wall or core to avoid jetting.
- To avoid trapped gas, the resin flow from the gates should direct air toward the vents.
- Gates should be located to provide flow from thick to thin sections; to minimize weld lines; and away from impact or other stressed areas.
- To minimize jetting, splay and gate blush, the gates should be located at right angles to the runner (Figure 1-10).
- Direct gating to a cosmetic surface may cause surface imperfections.

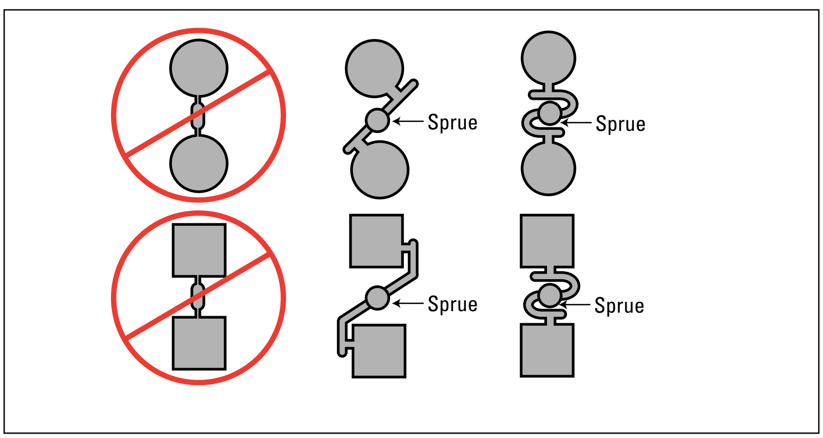


Figure 1-10. Runner to Gate (Indirect Approach).

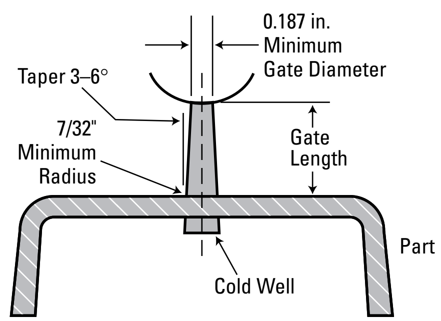
Direct Gating

Direct, center or sprue gates may be used on large, deep draw, or thick moldings where maximum injection pressure is necessary and long "gate open" times are required.

Caution: It is suggested that direct gates not be used in molding reinforced resins into rectangular shapes. Fiber orientation may result in part distortion.

In general, direct-sprue gates should be located in the thickest section of the part and designed with their diameters twice the thickness of the section but no greater than 1/2 inch (12.7 mm) in diameter. Keep the sprue as short as possible. Direct gating tends to increase the probability of gate blush or a sunburst effect particularly with glass reinforced materials (Figure 1-11).

Figure 1-11. Direct Sprue Gate.

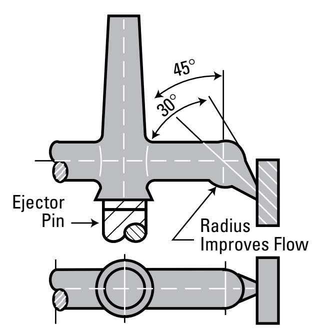


Tunnel Gating (Subgating)

This gate design permits automatic degating of a part from the runner system during ejection. A suggested gate approach is illustrated in Figure 1-12.

Approach to the tunnel gate should be fairly thick to prevent flow restriction, while the diameter of the orifice should measure no less than 0.050 inch (1.27 mm). A generous short approach off a full-round runner is suggested to prevent loss of injection pressure.

Figure 1-12. Tunnel Gate.



If the gate is not close to the sprue, it is suggested that an ejector pin be positioned near the gate as illustrated in Figure 1-13.

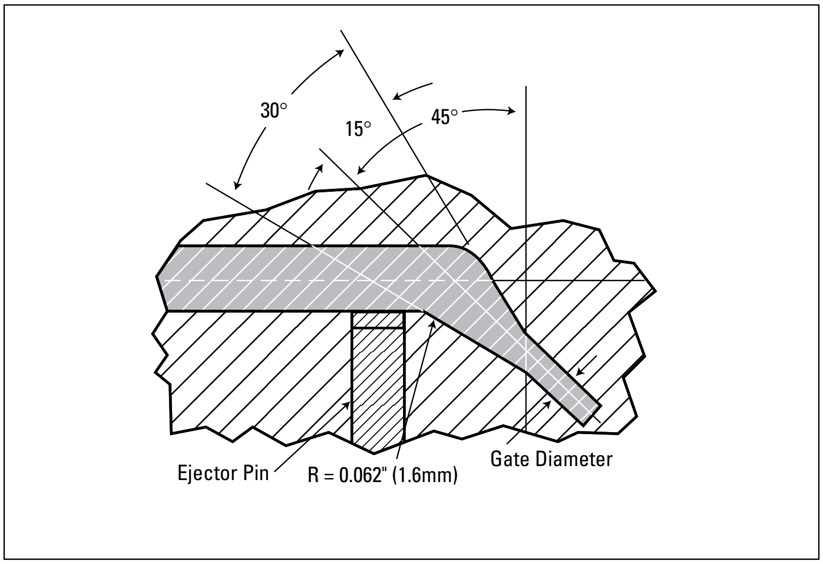


Figure 1-13. Tunnel Gate.

Pin Point Gating

Pin point gating (Figure 1-14) is used for three-plate molding. Diameters of gates for three-plate molds should be between 0.050 and 0.100 inch (1.27 and 2.53 mm). A gate with no land is suggested to direct the break-off to ease automatic ejection from the runner system. If attempts are made to reduce gate vestige left from the mold design (vestige shown on Figure 1-14), impact performance may be reduced because notches are created where the gate separates from the part.

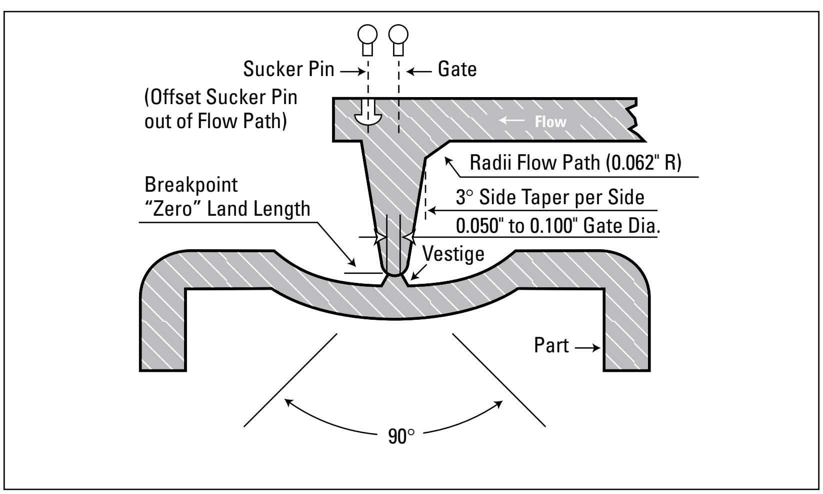
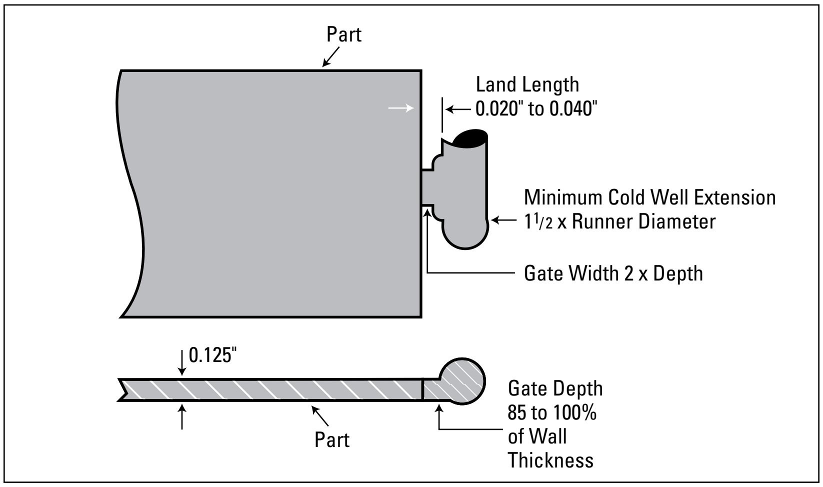


Figure 1-14. Pin Point Gate.

Edge Gating

Edge gates are the most commonly used gates in injection molding. For optimum resin flow, the height or thickness of the gate should generally be 85 to 100% of the wall thickness up to 0.125 inch (3.2 mm). The gate width should be 2 times the depth. A radius should be located at the junction of the molded part to prevent surface splay and to minimize molded-in stresses (Figure 1-15). A land length of 0.020 to 0.040 inch (0.50 to 1.01 mm) is suggested.

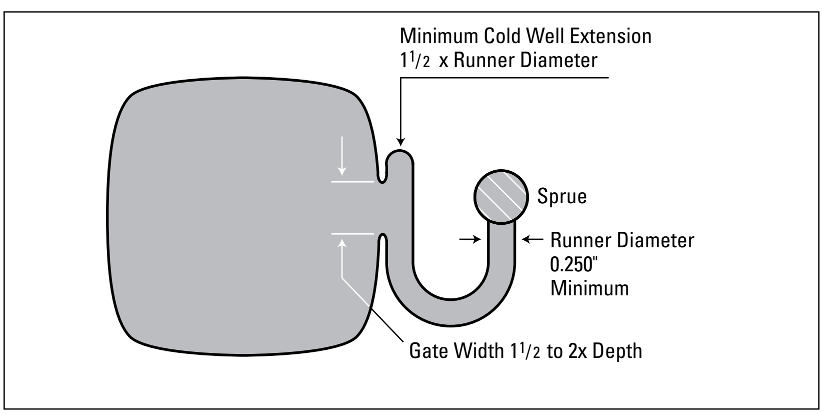
Figure 1-15. Edge Gating.



Modified Fan Gating

For flat, thin-walled sections, modified fan gating can minimize jetting and splay while significantly reducing high stresses caused by mold packing. The runner approach should be liberal and positioned 90° to the gate using as short a land length as possible (Figure 1-16). Smooth radii and transitions between runners and gates are suggested. Cold well extensions should be a minimum of 1-1/2 times the runner diameter. If gate blush or high stress persists, it is suggested that the cold well be extended. (Normally 2 to 3 times the runner diameter is sufficient.)

Figure 1-16. Modified Fan Gate.



Diaphragm Gating

Diaphragm or disk gates (Figure 1-17) are suggested for cylindrical parts requiring good concentricity and elimination of weld lines for strength.

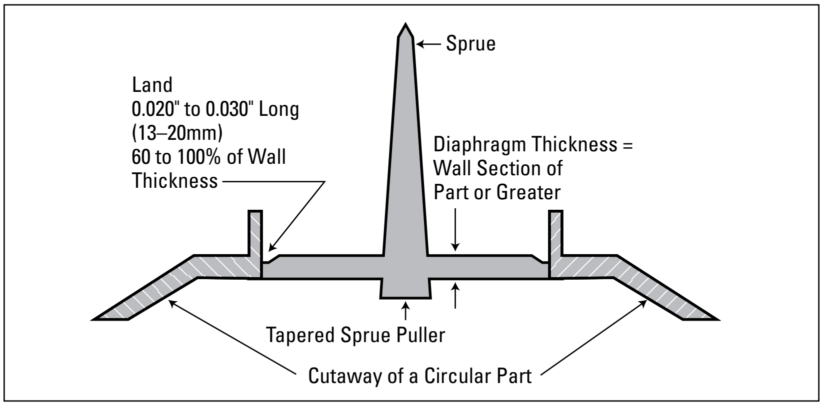


Figure 1-17. Diaphragm Gate.

Flash Gating

Flash gating (Figure 1-18) is an extension of fan gating. It is utilized to minimize warpage in flat designs or very large parts.

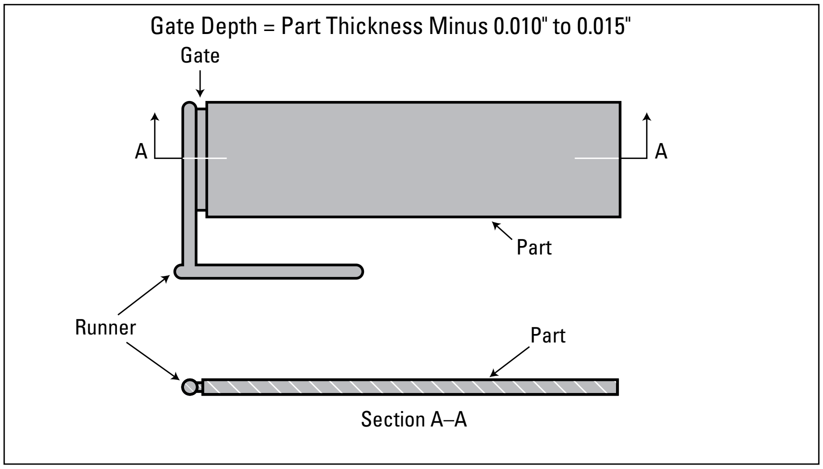
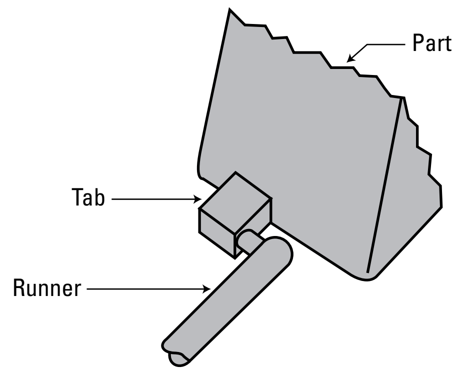


Figure 1-18. Flash Gate.

Tab Gating

If the indirect approach of a runner with a cold well cannot be used, the tab gate design can be used to reduce the effect of residual stresses and gate blush in the gate area (Figure 1-19).

Figure 1-19. Tab Gate.



Tolerances

Specifying only functional tolerances can reduce part cost and allow more economical mold construction. Every critical measurement should show the nominal dimension plus acceptable high and low limits. Excessively tight tolerances increase the cost of mold construction and often result in higher part cost.

Shrinkage

Typical mold shrinkage for most GE Plastics resins is low, uniform, and predictable. For general purpose unfilled amorphous resins it is in the range of 0.004 to 0.008 in./in. Glass reinforced or crystalline resins exhibit anisotropic shrinkage which must be considered in tool design. Crystalline resins also have high shrinkage rates due to their crystallinity.

Refer to data sheets for the appropriate range for each specific resin. Data sheet values allow for variations in part geometry and molding conditions.

Variations in wall thickness, packing pressure, mold and melt temperature, injection speed, along with others can have a large effect on the shrinkage of a material. Pressure-Volume-Temperature (PVT) data can be used in injection molding simulations to estimate the effect of processing on isotropic shrinkage. The PVT characteristics describe the changes in polymer volume which occur in response to variations in temperature and hydrostatic pressure. PVT data provide information regarding the compressibility and thermal expansion of the molten or solid polymer over the range of pressures and temperatures encountered in injection molding. See specific product information for specific volume as a function of temperature and pressure for specific resin grades.

Cavity Venting

When molding GE resins it is extremely important for the cavity to be vented effectively to allow air displaced by the melt to escape. Proper venting helps prevent "dieseling" (or super-heating of trapped air) and resulting burn marks at the end of the resin flow. This becomes even more critical in thin-wall parts and when using high injection speeds. Inadequate venting slows down the filling rate, resulting in incomplete mold fill. Vents must be provided at points of last fill and at weld lines (Figure 1-20). On parts with large surface areas, vents should be placed every 1 to 2 inches (25 to 50 mm) along the parting line.

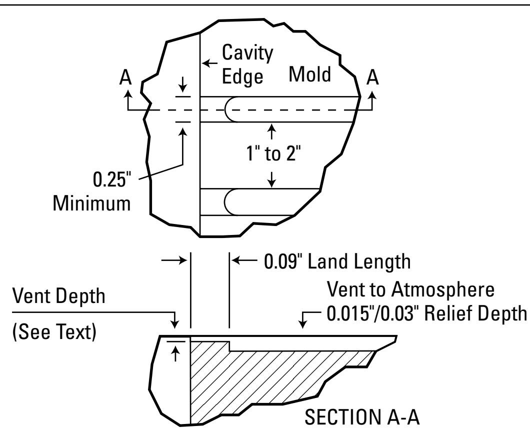


Figure 1-20. Parting Line Vent Detail.

The suggested land length is 0.09 inch (2.29mm). The vent relief to the atmosphere should be same width as the vents, with a suggested depth of 0.015 to 0.030 inch(0.38 to 0.76 mm) (Figure 1-20). Vent depths may vary from .0005 to .00075 for various crystalline resins to .003 for glass reinforced amorphous resins. See individual product sections in this publication for additional information.

Full perimeter venting is suggested whenever possible to avoid air entrapment at the parting line.

Some fill patterns may trap gasses in areas that cannot be vented at the parting line. To relive these gasses, vents can be machined on ejector pins, sleeve ejectors and moving cores to allow venting through the mold.

Venting of the runner cold slug wells can also improve melt flow into the cavity.

Mold Temperature Control

Adequate temperature control of the core and cavity surfaces is important for producing quality parts. Dual zone or separate controllers are required for independent temperature control of the two mold halves

Uniform mold temperature control is critical for maximization of cycles and control of part tolerances. It is generally advisable to maintain less than a 20˚F (-7˚C) differential in steel temperatures over a large cavity or core and less than 5˚F (-15˚C) for small ones. Tighter controls will generally provide greater processing latitude.

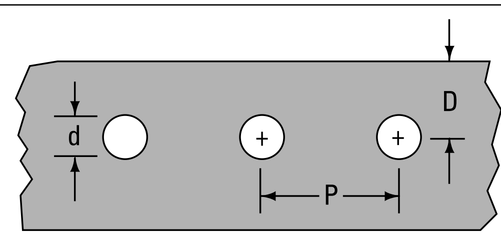
Typical mold cooling for parts molded in GE resins would incorporate 1/2 inch (12.7 mm) or larger cooling channels (1-1/2 inches to 2 inches (38 to 50.8 mm) apart and 1/2 inch (12.7 mm) below cavity and core surfaces

Other temperature control devices such as Logic Seal,® Thermal Pins® or bubblers can be used to aid temperature control in difficult-to-access areas.

Proper temperature control helps provide uniform heat across the tool surface. Therefore, looping of the cooling channels as shown in Figure 1-22 is not recommended practice. Water manifolds offer better control.

A large temperature differential across the mold surface creates different cooling rates and results in molded-in stresses in the parts. For the same reason it is generally not practical to maintain more than 40˚F (22˚C) difference between the core and cavity halves of the mold.

Figure 1-21. Cooling Channel Dimensions.



d – diameter	7/16"-9/16" (1.1113-1.4288 cm)
D – depth	d to 2d
P – pitch	3d to 5d

®Logic Seal is a Registered Trademark of Logic Devices, Inc.
®Thermal Pins is a Registered Trademark of Noren Products, Inc.

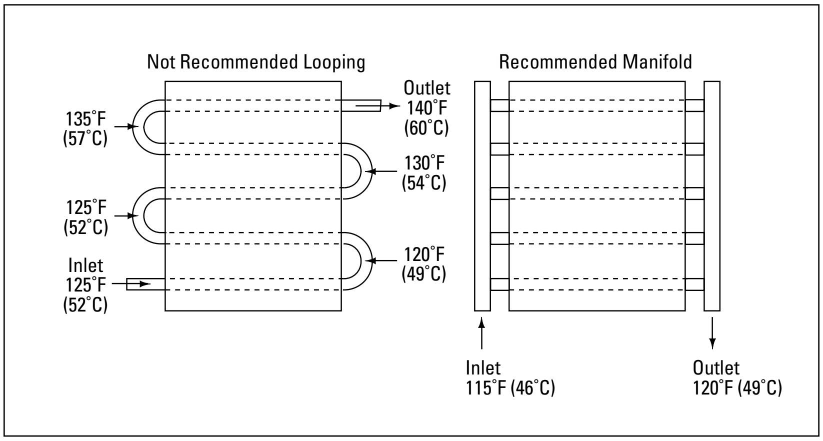


Figure 1-22. Cooling Channels.

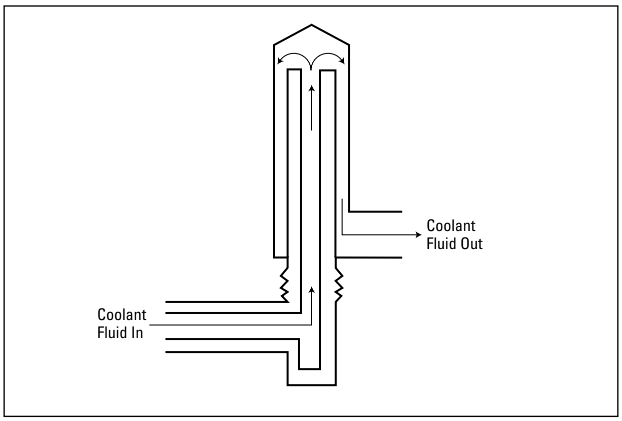


Figure 1-23. Bubbler (Cascading) for Spot Temperature Control.

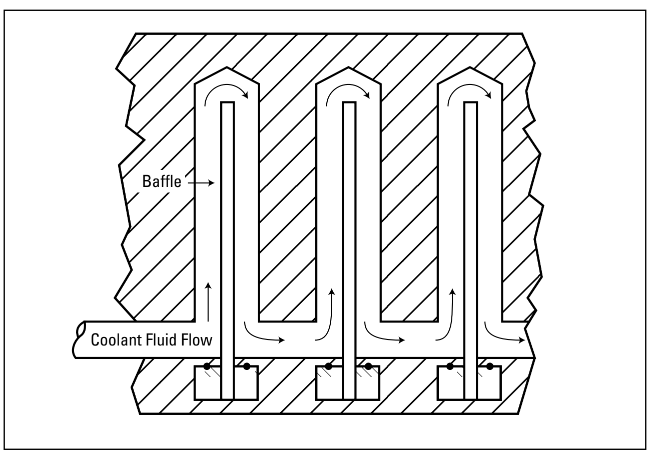


Figure 1-24. Baffles in Series (Single Core).

Additional information on mold surface temperatures can be found in the Typical Processing Parameter tables for each specific product line.

Figure 1-25. Heat Pipes – Thermal Pins.

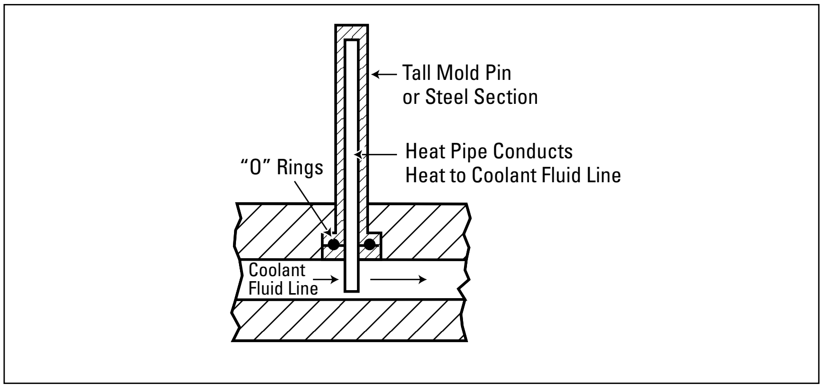


Figure 1-26. Coolant Fluid in Inserts.

Coolant in the core and cavity inserts and frame is preferred.

Coolant in the frame only is not suggested.

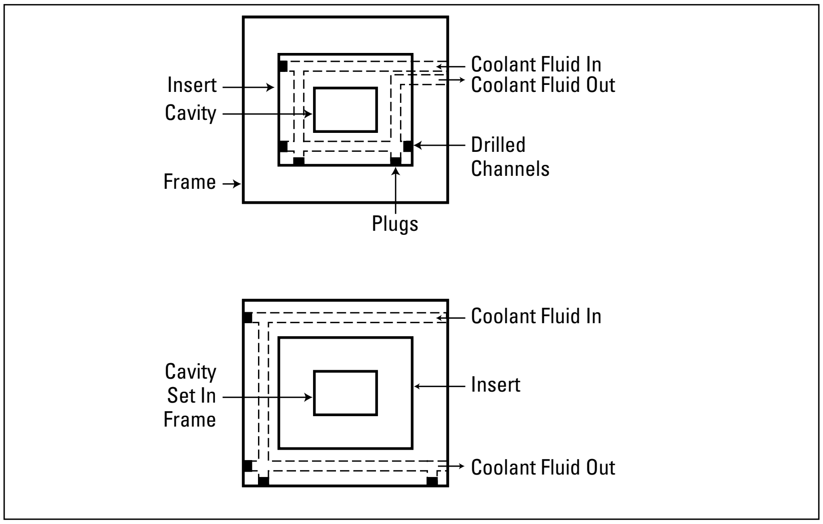
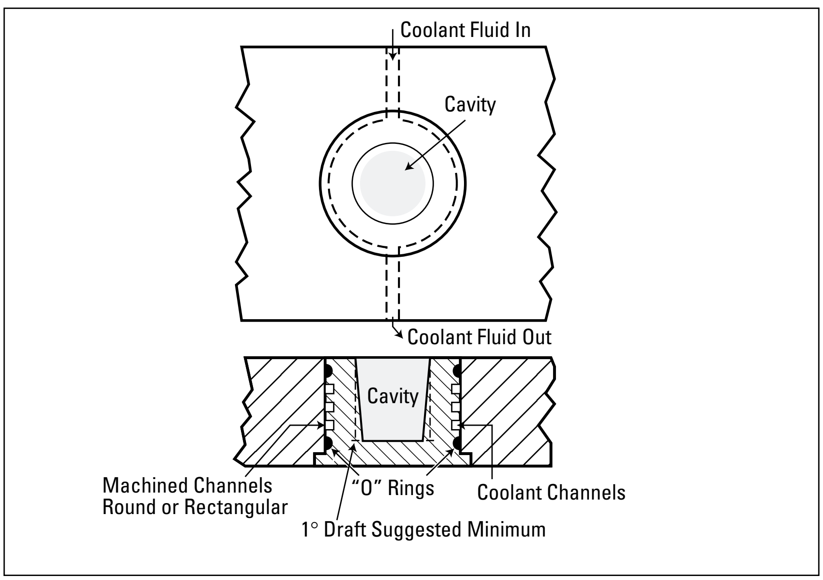


Figure 1-27. Coolant Fluid Lines Around Circular Inserts.



Mold Build-Up

When design permits, it is often desirable to incorporate an independent temperature control channel around the periphery and under the edge detail of the parts. This can allow the use of higher steel temperatures for the reduction of stress in the molded part edges (Figure 1-28).

Heated mold surfaces are normally needed when molding GE resins.

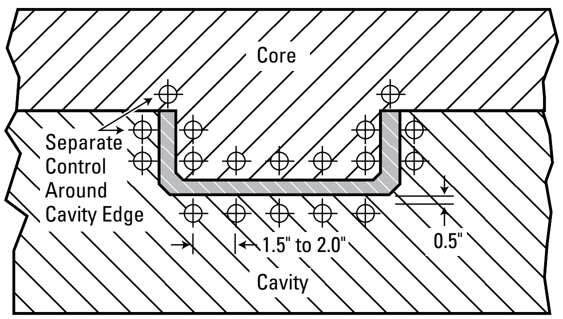


Figure 1-28. Cavity and Core Temperature Control.

Each part design must be reviewed for special cooling considerations.

Bubblers should be used for larger standing cores when conventional channeling is not possible (Figures 1-22 and 1-23). Bubblers may be put in series in a single core. If bubblers are used in individual cores, they should be parallel for individual circuits. Heat pipes can be used in smaller standing sections as well as chrome plated copper alloy inserts and pins (Figure 1-24).

The thermal conductivity of copper alloys can be as much as three times greater than steel, thus offering special advantages for part designs with difficult cooling problems.

Draft

Draft should be included on all surfaces in the line of draw of the die or a cam. Normally, the greater the draft angle, the easier it is to eject the part from the mold. Draft angles of 1˚ to 2˚ are preferred. In tight details or special fit requirements, a 1/2˚ minimum draft is advised.

In parts utilizing textured surfaces, additional draft is required. For textured walls in the line of draw of the mold, 1˚ additional draft for each 0.001 inch (0.03 mm) depth in texture is generally required.

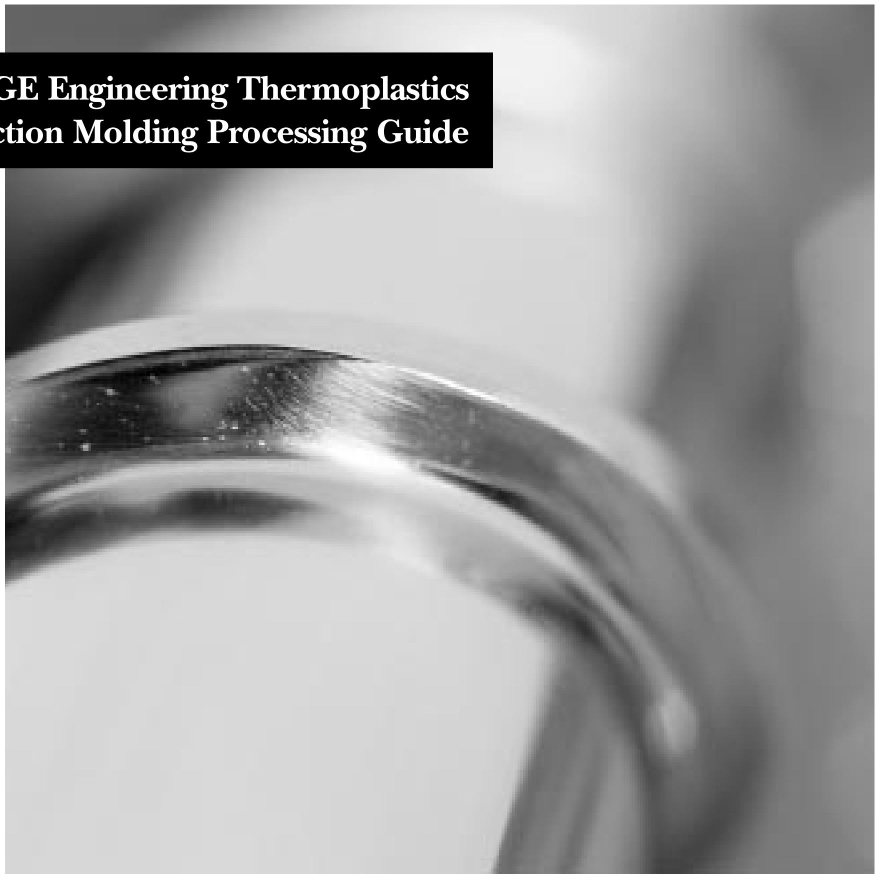
Part Ejection

Part ejection can be accomplished in several ways. The use of stripper plates is the most preferred method of part ejection because of the large contact area. The greater the part ejection contact area the less the chance for induced stress in the molded part, resulting in better dimensional integrity and reduced part damage. The use of stripper bars is another common method of part ejection that has a good part contact area.

Knockout pins are a common method of part ejection. For optimum results it is important to use an ample number of knockout pins. These pins should be designed with sufficient area to avoid compressing the resin surface and be located so that they do not induce stress in the part. The use of sleeve ejectors for part bosses is suggested.

External mold release agents are usually unnecessary for GE resins when molds are properly designed and polished. Draw-polishing is suggested for molds with thin ribs or minimal draft of vertical walls. In extreme cases, the use of various tool plating processes can aid in part ejection.

GE Plastics



GE Engineering Thermoplastics
Injection Molding Processing Guide

Injection Molding
Processing

Contents

Processing

Equipment

GE resins can be molded in standard injection molding machines. Reciprocating screw injection molding machines are preferred. This type of machine can deliver a homogeneous melt with more uniform temperature and flow characteristics throughout the shot. All flow paths should be streamlined to help eliminate material hang-up and degradation. High shear conditions should be avoided to help maintain proper temperature control.

Machine Selection

When determining the size of equipment to be used for molding a particular resin part, total shot weight and total projected area are the two basic factors to be considered.

Optimum results are generally obtained when the total shot weight (all cavities plus runners and sprues) is equal to 30 to 80% of the machine capacity. When using a hot runner melt delivery system, the volume of the material in the hot manifold should be included as an addition to the machine capacity for these calculations. Very small shots in a large barrel machine may create unnecessarily long resin residence times which may lead to resin degradation.

If it is necessary to mold at the high end of the temperature range, reduced residence time is usually required to reduce the possibility of material heat degradation. Therefore, for higher temperature molding requirements, it is suggested that the minimum shot size be greater than 60% of the machine capacity.

Once the total projected area of the complete shot (all cavity and runner areas subjected to injection pressure) has been determined, 3 to 5 tons of clamp force should be provided for each square inch of projected area to avoid flashing of the part. Wall thickness, flow length and molding conditions will determine the actual tonnage required. For example: thin wall LEXAN resin parts may require 6-8 tons/in^2.

Specific information on tonnage requirements for each resin family are presented in the special sections of this publication for each product line.

Barrel Selections and Screw Design Considerations

Conventional materials of construction for compatible screws and barrels are acceptable for processing GE resins. The use of bimetallic barrels is highly suggested.

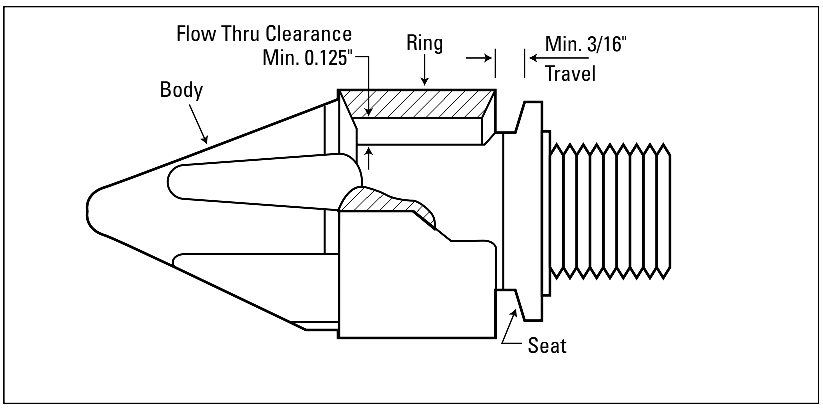
Depending on screw diameter, a compression ratio of about 2.5:1 with a length to diameter ratio of 20:1 is preferred. A short feed zone (5 flights) and a long compression zone (11 flights) with a gradual constant taper leading to a short metering zone (4 flights) is also suggested.

The compression should be accomplished over a gradual and constant taper since sharp transitions can result in excessive shear and material degradation. When specific screw selection is not possible, general purpose screws with length to diameter ratios from 18:1 through 24:1 and compression ratios from 2.0:1 to 3.0:1 have been used successfully. Vented barrels are not typically suggested for processing GE Plastics resins but could work for CYCOLAC® ABS resins and NORYL® modified PPO® resins in some limited applications.

The non-return valve should be of the sliding check ring type (Figure 2-1). Flow-through clearances of at least 80% of the cross-section of the flow area in the metering zone of the screw are usually necessary. The check ring travel should be at least 3/16 inch (4.76 mm) for small diameter screws [2 1/2 inch (63.5 mm) diameter or less]. Larger screws may require longer travel to provide the necessary flow-through area.

Ball check type screw tips are not suggested because they can cause degradation of the resin due to excessive shear and dead spots.

Figure 2-1. Non-Return Valve.



Nozzle Design

The nozzle opening should be a minimum 3/16 inch (4.76 mm) with 1/4 inch (6.4 mm) and 5/16 inch (7.9 mm) preferred. For parts 12 lbs. and up, a nozzle orifice of 3/8 inch (9.5 mm) (or greater) is suggested.

Land lengths should be no longer than 3/16 inch (4.76 mm). The nozzle bore should be 0.5 inch (12.7 mm) minimum.

For best results, the nozzle opening should typically be kept 1/32 inch (0.8 mm) smaller than the "O" or orifice dimension on the sprue bushing.

Accurate heat control and full heater band coverage is necessary for maintaining part appearance.

If extended nozzles are required, complete coverage with heater bands is essential to help maintain good temperature control. A separate control is suggested for the heater band at the tip of the nozzle. This is to help maintain proper temperature without overheating the main body of the nozzle. Heater bands that are hooked up to the same power source must be of the same wattage. If different watt values are used, separate voltage controls and temperature monitors must be used.

Specialty nozzles such as static mixers, shut-off nozzles and screen pack filters are not suggested, as they typically have sharp corners and areas of high pressure drop that can cause a variety of molding problems. The configuration of a general purpose nozzle is shown in Figure 2-2. Shut-off nozzles are recommended for structural foam and gas assist molding processing.

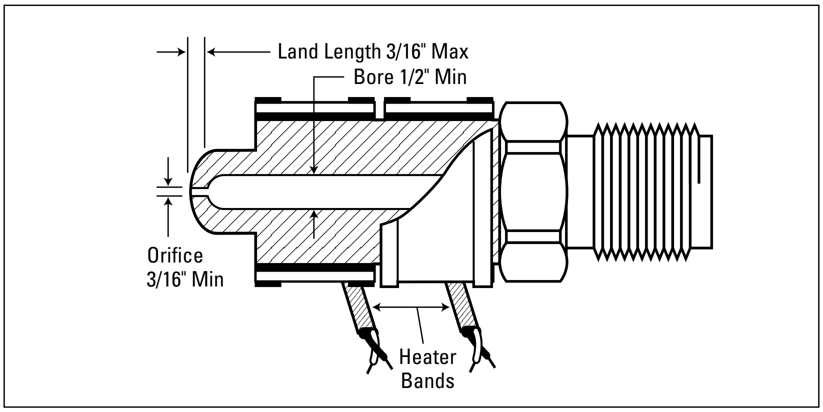


Figure 2-2. General Purpose Nozzle.

Drying

GE resins will absorb a small amount of water from the atmosphere after compounding and prior to processing. The amount absorbed will depend on the resin environmental conditions, and on the temperature and humidity of the storage area. Consistent drying of the resin increases the ability to stabilize processing parameters. Consistent tight processing parameters result in improved productivity by increasing part-to-part consistency and producing tougher parts. In order to reach the optimum performance of molded parts and to minimize the possibility of degradation, some grades of GE resins must be dried before processing. The required moisture level can usually be reached by pre-drying resins at the suggested time and temperature for each grade.

When using oven dryers, the resin should be spread in trays to a depth of approximately one inch. For large pellet size (regrind) or glass filled materials, the residence time should be increased to 4 to 6 hours. To avoid excessive heat history, it is suggested that the material be dried no longer than its maximum suggested time. It is suggested that the ovens be connected to a dehumidified air supply.

Processing

The hopper and any open areas of the feed mechanism should be covered to protect the dried pellets from room atmosphere. If a hopper dryer is not available, only a sufficient quantity (pellets to be used within 15 minutes) of dried, heated pellets should be removed from the oven and placed in the hopper at one time. The length of exposure to ambient atmosphere which the dried resin can withstand before a significant or potentially harmful amount of moisture is absorbed ranges from 15 minutes to several hours depending on relative humidity.

Where hopper dryers are available, oven drying can also be helpful to dry a quantity of resin for start-up. The hopper dryer should be preheated to the suggested drying temperature before the pellets are loaded in. Air entering the hopper should be at the suggested drying temperature and have a flow of 1.0 cfm for every pound per hour of use.

Drying Specialty Resins

Other drying parameters may apply to more recently developed GE resin grades, either in their virgin state or as regrind.

Prior to running new GE resins, refer to data sheets or contact a GE representative to insure that proper processing procedures are known. Call (800) 845-0600 to request technical assistance and product literature.

Drying Equipment

To avoid cross contamination, the dryers and material transfer system must be clean.

A closed loop, dehumidifying, recirculating hot air hopper dryer is suggested for drying GE resins (Figure 2-3).This system utilizes rechargeable desiccant cartridges to provide dry air (Figure 2-4). A correctly designed dryer and hopper provide a steady flow of dry pellets to the intake of the molding machine.

The dehumidified hot air flows through a hopper containing a spreader designed to prevent channel flow of the pellets and the air (Figure 2-3). If the spreader is not in place, proper air flow and hopper residence time may not be achieved.

The hopper capacity should be sized to provide the required residence time for the resin being molded. For example, a molding machine with a through-put of 100 pounds per hour would have a 400-pound hopper capacity to meet 3 to 4 hour drying time requirements.

The required drying temperature should be monitored at the input of the hopper (Figure 2-3). The dew point of the air at the input of the hopper should be -20˚F (-29˚C) to -40˚F (-40˚C) or lower.

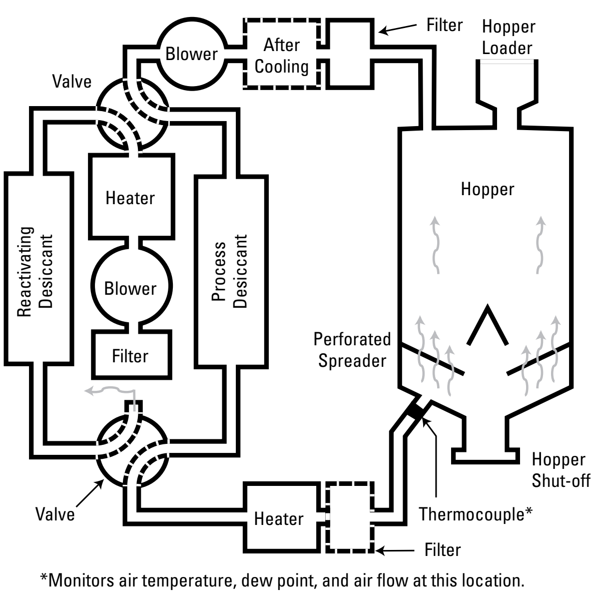
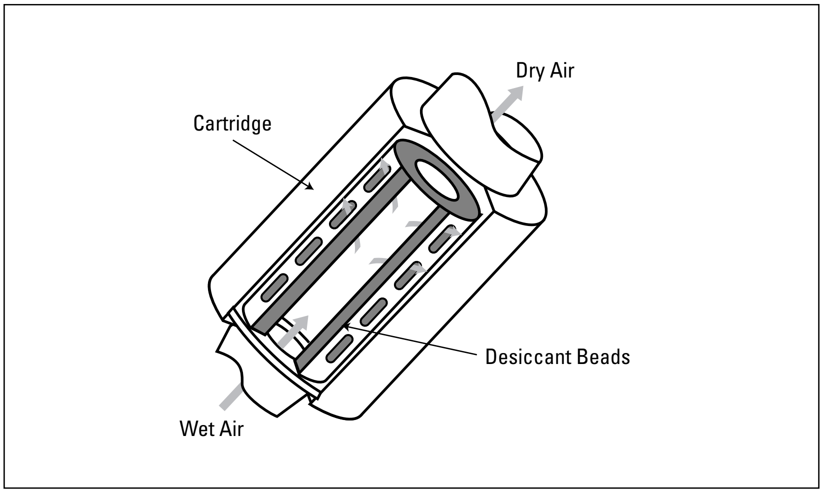


Figure 2-3. Schematic of a Typical Desiccant Dryer.

Figure 2-4. Desiccant Dryer.



Air flow volume should be one cubic foot of air per minute per pound of material processed per hour; the minimum air flow to process 100 pounds of resin per hour would be 100 cfm. Most desiccant dryers (Figure 2-4) have a filter system on the return air side to help eliminate contamination of the desiccant beds. These filters should be checked periodically to maintain the air flow for which the dryer was designed. Clogged filters cause a drop in air flow resulting in incomplete drying of the resin. If the resin is not properly dried, end use performance properties can be sacrificed.

If dryer systems with permanently installed desiccant beds have not been used recently, it is suggested that the system be dry cycled at approximately 150˚F (66˚C) for sufficient time to allow the reactivation of all desiccant beds.

Set the drying temperatures to values as suggested on the resin data sheets of the grade being processed (see separate product sections). After the drying temperature has stabilized at rated air flow, check the temperature at the heated dry air input of the dryer with a calibrated pyrometer or thermometer. When monitoring the air temperature, the temperature swing should be no more than plus or minus 10˚F (7˚C) from the suggested drying temperature.

Molding Conditions

As a general guideline, many standard grades of GE resins are molded at different temperatures. Specific information for each GE resin family is presented in the special sections in this guide for each product line.

Melt Temperature

Mold temperatures are important in determining final part finish and molded-in stress levels. Cold molds are more difficult to fill, necessitating high injection pressure and melt temperature. Heated molds generally produce a part with a better finish and lower molded-in stress. Because of the high heat distortion temperatures of engineering thermoplastic resins, parts are ejected easily at higher temperatures.

Most GE resins have outstanding thermal stability within the suggested melt temperature range. As a general rule, residence time should be as short as possible when molding near the maximum suggested melt temperature.

When processing near, or at, the upper limit of the melt range, the shot weight should approach 80 to 90% of the cylinder capacity of the machine. If the cylinder temperature exceeds the upper limit of the suggested melt range, thermal degradation of the resin and loss of physical properties may result.

A relatively small increase in screw speed (RPM) or back pressure can result in a dramatic increase in melt temperature with no change in controller set point. It is suggested that melt temperatures be measured using hand-held pyrometers. These measures should be taken on the thermoplastic melts after the machine is on cycle.

Increased melt temperatures with amorphous materials reduce viscosity and increase resin flow, thus providing for longer flow for thin wall sections and help produce lower residual stress. When higher melt temperatures are used, reduced barrel/residence times are needed to minimize any degradation of the resin.

The fastest fill speed possible provides longer flow, fills thinner wall sections, and helps to create a better surface finish. Slower fill is suggested for sprue-gated and edge-gated parts to help prevent gate blush, splay and jetting. In thick wall parts [0.2 inches (5.06 mm) and up] slow fill helps reduce sinks and voids.

For specific information on melt temperatures for each resin grade refer to the Typical Processing Parameters tables located in special brochures for each product family. (See page ii)

Engineered thermoplastics should not be left at elevated temperatures for prolonged periods of time without occasional purging.

Mold Temperature

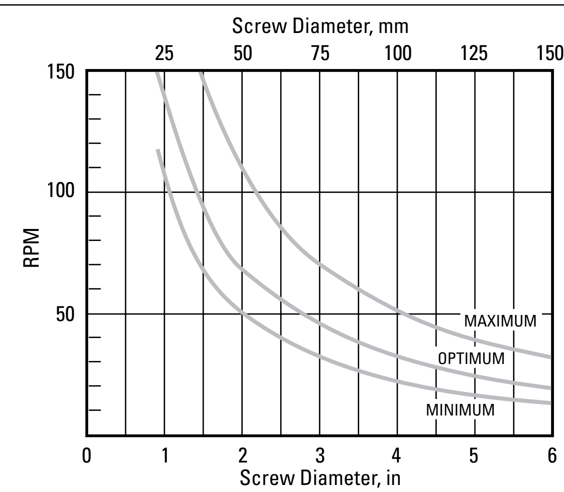
The midpoint of the suggested range will generally give good results with respect to part appearance and cycle time. Higher mold temperatures can result in better flow, stronger knitlines and lower molded-in stresses. Using lower than the suggested mold temperatures will generally result in higher molded-in stresses and compromise part integrity.

Screw Speed

Screw speeds (RPM) should be adjusted to permit screw rotation during the entire cooling cycle without delaying the overall cycle (see Figure 2-5). Low screw speeds will help reduce glass fiber damage during plastication when molding reinforced grades.

Suggested screw speed is dependent on screw diameter. Optimum linear velocity of screw O.D. is typically 8 inches (202.4 mm) per second. RPM = screw diameter × π divided into the optimum linear velocity of 8 inches (202.4 mm) per second × 60. For example, for a 3 inch (75.9 mm) diameter screw: 3 (screw Dia.) × 3.1416 = 9.4248 divided into 8 inches (202.4 mm) per second (optimum linear velocity) × 60 = 51 RPM.

Figure 2-5. Screw Speeds Suggestions for GE Engineering Resins.



Back Pressure

A back pressure of 50 to 100 psi (0.35 to 0.7 MPa) is suggested to help ensure a homogeneous melt and maintain consistent shot size. Higher back pressures used to improve melt mixing result in higher melt temperatures and possibly degradation due to excessive shearing of the resin.

Shot Size

The suggested shot size is 30 to 80% of the machine capacity. It should be noted that the displacement size of the barrel is rated in ounces of impact of styrene with a specific gravity of about 1, and should be adjusted to account for materials with different specific gravities.

Ram Speed

When selecting injection speed, careful consideration must be given to adequate mold venting, resin melt temperature and injection pressure, along with the potential for jetting.

The fastest fill speed possible provides longer flow, fills thinner wall sections, and helps to create better surface finish. In thick parts, slow fill helps reduce voids. Thin-wall sections below 0.06 inch (1.52 mm) require fast ram speeds in order to fill the cavity and produce high knitline strength. The fill rate of thick sections may be reduced to aid packing when filling through restricted gates.

Programmed injection is suggested for parts with small gates (pin gates and subgates). A slow injection rate can be used at the start to help eliminate gate blush, jetting, and burning of the material. Once underway, speed can be increased to fill the part. Maintain constant melt front velocity, and slow near end of fill to optimize part quality.

Injection Pressure

The actual injection pressure will depend on variables such as melt temperature, mold temperature, part geometry, wall thickness, flow length, and other mold and equipment considerations. Generally, the lowest pressures which provide the desired properties, appearance, and molding cycle are preferred.

Ideally, injection pressure should vary to maintain the intended ram speed. This is called “velocity controlled injection” and provides the molder a much more robust and respectable process than controlling injection pressure during fill.

Pack/Hold Pressure

Holding pressures from 40 to 80% of the injection pressure are adequate for normal requirements.

Processing

Downtime

When it becomes necessary to stop molding GE resins, the following steps are suggested:

- Maintain cylinder temperatures for interruption up to 15 minutes.
- Decrease cylinder temperature by 100˚F (56˚C) for periods from 15 minutes to 2 hours.
- Reduce further to 350˚F (177˚C) for interruptions from 2 to 8 hours.
- Purge out barrel and shut of heat for periods longer than 8 hours.
- For glass reinforced resins purge out barrel if downtime is going to be longer than 30 minutes.

For prolonged shut-down or storage, molds should be cleaned and coated with neutralizer and dehydrator containing a rust inhibitor.

Purging

Purging operations should be done with the safety shield in the down position to cover the barrel end and nozzle to contain possible splatter or blow-back of the molten plastic.

Adequate purging can usually be accomplished with the nozzle in place or removed. Removal of the nozzle will allow complete cleaning and inspection for roughness or foreign objects.

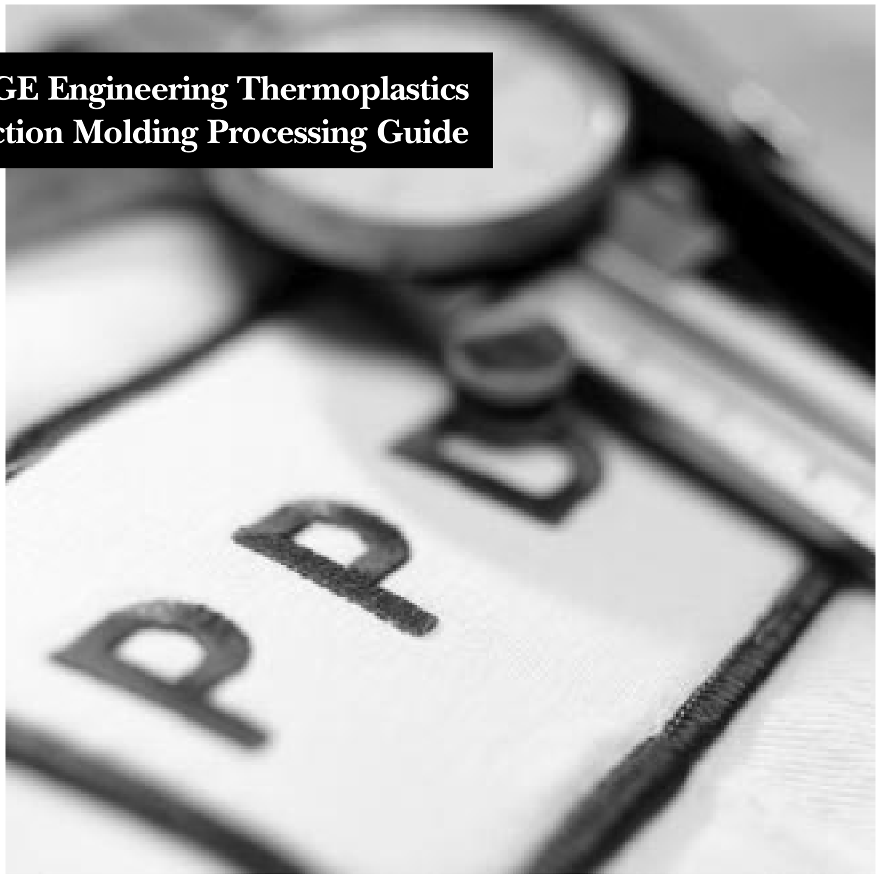
Polystyrene and reground cast acrylic are effective purging materials for GE resins. Purging should be done within the melt temperature range for that particular grade of resin. It is important to have proper ventilation during the purging procedures. Timing for purging is resin specific. Due to their high processing temperatures, ULTEM® resins have additional considerations to consider when purging. See ULTEM resin processing brochure for specifics.

Regrind

If the application permits the use of regrind, reground sprues, runners, and non-degraded parts may be added to the virgin pellets up to a level of 25%. Grinder screen sizes should be 5/16 to 3/8 inch (7.9 to 9.5 mm). If a smaller size is used, too many fines could be generated, creating molding problems such as streaking and burning. It is important to keep the ground parts clean and to avoid contamination from other materials. Drying time should be increased since regrind will not be the same size as virgin pellets, and therefore water diffusion will be different. Regrind utilization may produce some effect on color. Actual regrind usage should be determined for each individual application.

GE Plastics

GE Engineering Thermoplastics
Injection Molding Processing Guide



Injection Molding Troubleshooting Guide

Troubleshooting Tips

Inasmuch as General Electric Company has no control over the use to which others may put its engineering thermoplastics, it does not guarantee that the same results as those described herein will be obtained. Nor does General Electric Company guarantee the effectiveness or safety of any possible or suggested design for articles of manufacture as illustrated herein by any photographs, technical drawings and the like.

Each user of GE engineering thermoplastics or design, or both, should make his own tests to determine the suitability of the material, or any material for the design, as well as the suitability of the material or design, or both, for his own particular use.

Statements concerning possible suggested uses of engineering thermoplastics or designs described herein are not to be construed as constituting a license under any General Electric patent covering such use, or as recommendations for use of such materials or designs in the infringement of any patent.

The information present below describes some commonly encountered problems associated with injection molding engineering thermoplastics with suggestions and steps that can be taken to address them. Because of the many variables involved in injection molding, there can be no certainty that a particular step, or group of steps, will solve a particular problem.

If problems are still encountered after the potential solutions described here are tried, contact your GE Plastics Technical Development Engineer for more information.

PROBLEM: BLACK SPECS

Small black area (spots) inside the material, mostly present in transparent LEXAN and ULTEM resin grades.

POSSIBLE SOLUTIONS

- If previously molding other types of polymers, purge other polymer out of barrel completely. Other polymers tend to degrade at the high ULTEM resin processing temperatures
- Always purge when not running for extended periods of time
- Decrease nozzle temperature
- Check temperature at feed section, low temperatures may cause mechanical degradation especially with a high screw speed or high back pressure
- Check heater band and thermocouple location. The heater band may be heating to a higher temperature than the thermocouple is measuring
- Check nozzle tip, nozzle adapter, and end-cap for hang-up areas
- Check the type of screw used, screws with long transition sections are typically suggested for ULTEM resins. Short transition sections could result in mechanical degradation.
- Perform routine screw maintenance
- Check the radius between the screw root and the flighter. A generous radius is usually suggested to prevent material hang-up behind the flights.
- If material degradation is suspect, lower material temperature by:
  - a. Lowering cylinder temperature
  - b. Decreasing screw speed
  - c. Lowering the back pressure

PROBLEM: BRITTLENESS

The molded product is failing prematurely either after molding, during testing, or during normal usage.

POSSIBLE SOLUTIONS

- Dry resin properly (moisture greater than 0.02% can cause loss of properties)
- Improve weld line strength (see weldlines)
- Check melt temperature with pyrometer

PROBLEM: BURN MARKS

They are usually caused by overheating the material due to entrapped air (diesel effect): this causes the darkening in color.

POSSIBLE SOLUTIONS

- Decrease injection speed
- Decrease booster time
- Decrease injection pressure
- Check venting channels for dirt
- Use programmed injection speed
- Improve venting of tool. (Add vents at burn locations)
- Alter position of gate and/or increase gate size

PROBLEM: DISCOLORATION

The appearance of a non-uniform color distribution in the molding.

POSSIBLE SOLUTIONS

- Purge heating cylinder
- Check melt temperature with pyrometer
- Lower material temperature by:
  - a. Lowering cylinder temperature
  - b. Decreasing screw speed
  - c. Lowering the back pressure
- Increase back pressure to improve melt homogeneity
- Lower nozzle temperature
- Shorten overall cycle
- Check hopper and feed zones for contaminants
- Check for proper cooling of ram and feed zone
- Provide additional vents in mold
- Move mold to a smaller shot size press to reduce residence time
- Check hot manifold for dead spots

Injection Molding

PROBLEM: GLOSS

Surface gloss is resin and processing dependent.

POSSIBLE SOLUTIONS

- Increase mold temperature
- Increase melt temperature
- Increase injection speed
- Increase injection pressure
- Check surface of the mold for polish
- Clean vents
- Increase venting

PROBLEM: JETTING/WORMING

It shows as a serpentine line on the part surface emanating from the gate

POSSIBLE SOLUTIONS

- Decrease injection speed
- Increase resin temperature
  - a. Increase cylinder temperature
  - b. Increase screw speed (unfilled)
  - c. Increase back pressure (unfilled)
- Increase nozzle or zone temperature
- Increase gate size
- Decrease gate land length
- Modify gate location or angle: directly into wall or pin
- Avoid gating at thick section

PROBLEM: SINK MARKS

Visible defects resulting from insufficient cooling before removal from the mold. A heavy rib intersecting a thin wall may show up as sink marks: these are very difficult to eliminate by varying processing conditions

POSSIBLE SOLUTIONS

- Follow rib design guidelines
- Increase injection speed to maximum range
- Increase injection hold time
- Increase injection pressure
- Reduce melt temperature
- Reduce mold temperature
- Enlarge and/or add vents to mold parting line
- Increase size of sprue and/or runners
- Increase gate size and reduce gate land length
- Relocate gate next to heavy or thicker areas
- Core out heavy wall sections where possible

PROBLEM: SPLAY MARKS

The result of (a) moisture on the pellets which should be removed under recommended drying times and temperatures (b) products of degradation due to overheating (c) residual non-aqueous volatiles in material.(a) and (c) will typically produce fine lines emanating from the gate all over the part whereas (b) will show up as coarse lines, lumps in sections of the molded parts.

POSSIBLE SOLUTIONS

- Dry resin properly. Generally, excess moisture may cause splay
- Lower nozzle temperature
- Lower material temperature by
  - a. Lowering cylinder temperature
  - b. Decreasing screw speed
  - c. Lowering the back pressure
- Decreasing injection speed
- Increase or decrease mold temperature
- Shorten or eliminate screw decompression
- Shorten overall cycle
- Check for contamination (e.g. water or oil leaking into mold cavity)
- Move mold to smaller shot size press
- Check for drooling
- Add “screw decompression” to molding cycle
- Open gates
- Check hot runner system
- Increase nozzle orifice
- Increase sprue and runner size

PROBLEM: STICKING IN CAVITY/CORE

At the end of cycle, the molding does not release from the mold but sticks on the core or cavity

POSSIBLE SOLUTIONS

- Decrease injection pressure
- Decrease hold time and pressure
- Decrease booster time
- Adjust feed for constant cushion
- Increase or decrease mold closed (cooling) time
- Is part geometry dependent?
- Adjust the cavity/core temperatures to a 20˚ differential between mold halves
- Decrease cylinder and nozzle temperature
- Check mold for undercuts and/or sufficient draft

PROBLEM: STICKING IN SPRUE BUSHING

At end of cycle, the sprue does not release from the mold but sticks in the sprue bushing

POSSIBLE SOLUTIONS

- Decrease injection pressure
- Increase injection hold time
- Decrease booster time
- Increase mold closed time
- Decrease mold temperature at sprue bushing
- Leave nozzle against mold: no pull back
- Raise nozzle temperature
- Check for incorrect seat between nozzle and sprue: size and alignment of holes in nozzle and sprue bushing
- Nozzle orifice should typically be 0.30" smaller in diameter than sprue bushing o-diameter
- Check polishing of sprue
- Provide more effective sprue puller
- Make sure sprue has enough draft angle for easy release
- If the sprue is stringing, increase or add screw decompression

PROBLEM: STRESS IN PART

These molded in stresses can result in part brittleness. Usually caused by highly oriented polymer flow.

POSSIBLE SOLUTIONS

- Increase mold temperature
- Decrease injection speed
- Increase melt temperature
- Decrease injection pressure
- Increase gate size
- Increase nozzle orifice diameter
- Add gates. Relocate gates

PROBLEM: VOIDS

Vacuole hollows (empty bubbles) in the molding, generally due to thermal shrinkage that draws material away from the fluid core of a part.

POSSIBLE SOLUTIONS

- Decrease injection speed to medium range
- Increase holding time
- Reduce melt temperature
- Increase mold temperature
- Increase gate size and reduce gate land length
- Increase nozzle size and/or runner system
- Redesign part to obtain equal wall section

PROBLEM: WARPAGE, PART DISTORTION

A dimensional distortion in the molded part usually bowing or bending of the part

POSSIBLE SOLUTIONS

- Set differential mold temperatures to counteract warpage due to part geometry
- Observe mold for uniform part ejection
- Check handling of parts after ejection from mold
- Increase injection hold time until gate freezes
- Increase mold closed time
- Increase or reduce injection pressure
- Increase mold temperature

- Use shrink fixtures and jigs to promote uniform cooling of the part
- Check gate locations and total number of gates to reduce orientation
- Increase gate dimensions
- Redesign part to equalize wall variation in molded part-thick and thin walls in the same part create differential shrinkage stresses
- Check cooling line layout, unbalanced cooling promotes warpage
- Relocate gates

PROBLEM: WELD LINES/KNIT LINES

These lines occur where two plastics flow fronts in the mold meet. The streams of plastic should be hot enough to fuse adequately. Weld lines are not just surface marks, but can be points of weakness: notches, stress raisers.

POSSIBLE SOLUTIONS

- Increase injection speed
- Increase injection pressure
- Increase injection hold time
- Raise the mold temperature
- Raise melt temperature by increasing cylinder temperatures
- Vent the cavity in the weld area
- Provide an overflow well next to the weld area
- Change gate location to alter flow pattern
- Reduce filler content

Suggested Do's and Don'ts

DO'S

1. Do dry resin at temperature suggested. MOISTURE CONTENT SHOULD BE BELOW 0.02%.
2. Do use adequate runners (full round) 1/4 to 3/8 inch (6.35 to 9.53 mm).
3. Do use proper size equipment for shot – 30 to 80% machine capacity.
4. Do vent mold cavity (0.002 to 0.003 inch (0.05 to 0.07 mm).
5. Do use proper screw equipment.
6. Do use straight bore nozzles.
7. Do use an appropriate mold temperature.
8. Do use moderate screw back pressure for proper feed – 50 to 100 lbs. gauge line pressure.
9. Do check non-return valve periodically for damages.
10. Do use the recommended screw speeds (25-75 rpm, dependent on screw diameter).

DON'TS

1. Don't use injection machine nozzle shut-off valve, except for when using structural foam or gas assist injection molding.
2. Don't use reverse taper nozzles.
3. Don't use small nozzle orifice – less than 3/16 inch (4.76 mm).
4. Don't use excessive back pressure.
5. Don't use excessive screw speeds.
6. Don't use long nozzle [3 inch (75.9 mm) or longer] without proper heat control.
7. Don't use pinpoint gates below 40 mils.
8. Don't use excessive resin temperature for the grade being molded.
9. Don't use same molding profile (heat and cycle) when changing resin grades. Each has its own conditions.
10. Don't use wet resin, nor resin that has been out of the dryer for longer than 20 minutes.

Notes

Notes

Literature

Design Guide

PBG-130

A comprehensive 380-page guide that provides in-depth information engineers can use to help design state-of-the-art equipment and components with GE engineering thermoplastics. Includes explanations of product groupings and grades, application development assistance, procedures and data relating to material selection, design assistance, prototyping and processing considerations, assembly and finishing details. Also included is a glossary of commonly used engineering terms that relate to plastics.

Product Guide

PBG-140I

Contains information designers and processors can use to help understand the characteristics of various polymer chemistries and GE Plastics resin families. Includes helpful information to select engineering thermoplastics for a specific application. Provides comprehensive property profiles and descriptive information on GE Plastics' entire product portfolio with data on testing methods utilized to evaluate engineering materials.

Injection Molding Processing Guide

PBG-135

Over 200-pages of information processors can use to help fabricate components using the injection molding process. Includes general information on converting engineering thermoplastics together with specific processing details and molding parameters for each GE resin family and grade. Additional processing publications containing information on thinwall, gas-assist, blow molding, extrusion, thermoforming, and structure foam processing techniques are also available.

The Weatherables™ resins

GED-WEA-105

Proven in the toughest environments, The Weatherables resin portfolio consists of a group of materials capable of delivering an exceptional list of benefits to outdoor applications — color-fast performance and appearance, corrosion resistance, eco-friendly attributes, long-term durability and the convenience of low, or no, maintenance. Specific grades of GELOY®, VALOX® and XENOY® resins make up this added-value line-up of materials.

Crystalline Products

VAL-500

Details of products that comprise GE Plastics' high-performance crystalline product portfolio. Includes property and application information on VALOX®, XENOY® and ENDURAN™ resins and a summary of applications and markets that utilize these materials.

The Internet

Complete access to over 3,000 pages of technical information is available on-line on the Internet. Users can navigate within GE Plastics' www.geplastics.com address to access the literature described above as well as GE Select™, an engineering database containing product data sheets for over 500 grades of GE engineering resins.

® GELOY, VALOX, and XENOY are Registered Trademarks of General Electric Company.
™ ENDURAN, The Weatherables and GE Select are Trademarks of General Electric Company.

Sales Offices

AMERICAS

United States

GE Plastics
One Plastics Avenue
Pittsfield, MA 01201 USA
Telephone: (413) 448-7110
Cable: GEPLASTICS

California

100 So. State College Blvd.
Brea, CA 92821
Telephone: (714) 255-7200

★ 4160 Hacienda Drive
Pleasanton, CA 94588
Telephone: (510) 734-0161

Georgia

★ Commercial Development Center
205 Scientific Drive
Norcross, GA 30092
Telephone: (770) 662-1000

Illinois

Suite 100, One Corporate Lakes
2525 Cabot Drive, Lisle, IL 60532
Telephone: (630) 505-2500

Massachusetts

★ One Plastics Avenue
Pittsfield, MA 01201
Telephone: (413) 448-7110

Michigan

P.O. Box 5011
Southfield, MI 48086-5011

★ 25900 Telegraph Road
Southfield, MI 48034
Telephone: (810) 351-8000

Ohio

Suite 660, 6000 Lombardo Center
Seven Hills, OH 44131
Telephone: (216) 524-2855

Texas

Suite 930, 5430 LBJ Freeway
Dallas, TX 75240
Telephone: (972) 458-0600

Puerto Rico

General Computer Building
P.O. Box 2010
Bayamon Puerto Rico 00960
Road 174, No. 101
Minillas Industrial Park
Bayamon Puerto Rico 00959
Telephone: 787-288-2340
Telefax: 787-288-2348

Brazil

GE Plastics South America S/A
Av. das Nacoes Unidas, 12995-20 andar
Brooklin Novo
04578-000 São Paulo, SP Brazil
Telephone: 55-11-5505 2800
Telefax: 55-11-5505 1757

★ Application Development Center

Canada

GE Plastics – Canada
General Electric Canada Inc.
2300 Meadowvale Blvd.
Mississauga, Ontario
L5N 5P9 Canada
Telephone: (905) 858-5700
Telefax: (905) 858-5798

Mexico

GE Plastics Mexico S.A. de C.V.
Av. Prolongación Reforma #490
4o. Piso
Colonia Santa Fe
01210 Mexico, D.F.
Telephone: 525-257-6060
Telefax: 525-257-6070

EUROPE

European Headquarters

★ General Electric Plastics B.V.
Plasticslaan 1
4612 PX Bergen op Zoom
The Netherlands
Telephone: (31) 164-292911
Telefax: (31) 164-292940

General Electric Plastics B.V.
P.O. Box 117
4600 AC Bergen op Zoom
The Netherlands
Telephone: (31) 164-292911
Telefax: (31) 164-291725

Austria

GE Plastics Austria
Pottendorfarstrasse 47
A-2700 Wiener Neustadt, Austria
Telephone: (43) 2622-39070
Telefax: (43) 2622-39047

France

★ General Electric Plastics France
S.a.r.L.
Z. I. de St. Guénault
Boite Postale No. 67
F-91002 Evry/Cedex, France
Telephone: (33) 1 60796900
Telefax: (33) 1 60796922

Germany

★ General Electric Plastics GmbH
Eisenstraße 5
65428 Rüsselsheim
Postfach 1364-65402
Rüsselsheim, Germany
Telephone: (49) 6142 6010
Telefax: (49) 6142 65746

India

★ GE Plastics India Limited
405-B, Sector 20
Udyog Vihar Phase III
Gurgaon 122016 (Haryana)
Telephone: 91-124-341-801
Telefax: 91-124-341-817

Italy

★ GE Plastics Italia S.p.A.
Viale Brianza 181
20092 Cinisello Balsamo
Milano, Italy
Telephone: (39) 2 61834.301
Telefax: (39) 2 61834.305

Spain

★ GE Plastics Iberica, S.A.
Avinguda Diagonal 652-656
08034 Barcelona, Spain
Telephone: (34) 3 252 16 00
Telefax: (34) 3 280 26 19

United Kingdom

★ GE Plastics Ltd.
Old Hall Road
Sale, Cheshire M33 2HG
United Kingdom
Telephone: (44) 161-905-5000
Telefax: (44) 161-905-5106

PACIFIC

Pacific Headquarters

GE Plastics Pacific Pte. Ltd.
#09-00 GE Tower
240 Tanjong Pagar Road
Singapore 0208
Telephone: (65) 220-7022
Telefax: (65) 326-3290

Australia

★ GE Plastics (Australia) Pty. Ltd.
175 Hammond Road
Dandenong, Victoria 3175
Australia
Telephone: (61) 3 9703 7200
Telefax: (61) 3 9794 8563

GE Plastics (Australia) Pty. Ltd.
57/2 O'Connell Street
Parramatta, New South Wales 2150
Australia
Telephone: (61) 2 9689 3888
Telefax: (61) 2 9689 3530

GE Plastics (Australia) Pty. Ltd.
Legal and General Building
206 Greenhill Road
Eastwood, South Australia 5063
Australia
Telephone: (61) 8 8272 5044
Telefax: (61) 8 8272 2479

China

GE Plastics – Beijing
General Electric (USA) China
Company, Ltd.
3rd Floor, CITIC Building
No. 19 Jian Guo Men Wai Ave.
Beijing 100004, P.R. China
Telephone: 86-10-6500-6438
Telefax: 86-10-6500-7476

GE Plastics – Shanghai
GE China Company
9th Floor, Shartex Center
88 Zunyi Road (S)
Shanghai 200335, P.R. China
Telephone: 86-21-6270-9623
Telefax: 86-21-6270-9973

Hong Kong

GE Plastics Hong Kong Limited
Room 1008, Tower I, The Gateway
25 Canton Road, Tsimshatsui
Kowloon, Hong Kong
Telephone: 853-2629-0853
Telefax: 853-2629-0804

Indonesia

GE Plastics – Indonesia
KH Mas Mansyur Kav. 126
Jakarta 10220, Indonesia
Telephone: (62) 21-574-4980
Telefax: (62) 21-574-7101

Japan

★ GE Plastics Japan, Ltd.
1015 Aza Sumiyakizawa
Hodozawa, Gotemba-shi
Shizuoka 412, Japan
Telephone: (81) 550-89-2323

GE Plastics Japan, Ltd.
Tokyo Office
Nihombashi Hamacho Park Bldg.
2-35-4, Nihombashi-Hamacho
Chuo-ku, Tokyo 103, Japan
Telephone: (81) 3 5695-4861
Telefax: (81) 3 5695-4859

Korea

★ GE (USA) Plastics Korea Co., Ltd.
231-8, Nonhyundong Kangnamku
Seoul 135-010, Korea
Telephone: (82) 2 510-6250/6000
Telefax: (82) 2 510-6666/6224

Singapore

GE Plastics – Singapore
Sales & Marketing Office
80 Anson Rd., #38-00 IBM Towers
Singapore 079907
Telephone: (65) 223-7022
Telefax: (65) 223-7033

Taiwan

GE Plastics, Taiwan
8F-1, 35 Min Chuan E. Road Sec. 3
Taipei, Taiwan, R.O.C.
Telephone: (886) 2 509-2124
Telefax: (886) 2 509-1625

Thailand

GE Plastics – Thailand
21st Floor Thaniya Plaza Bldg.
52 Silom Road
Bangkok 10500, Thailand
Telephone: (66) 2-2312323
Telefax: (66) 2-2312322

GE Plastics
We bring good things to life.

General Electric Company
One Plastics Avenue
Pittsfield, MA 01201
800.845.0600
http: / / www.geplastics.com

PBG-135 (1/98) CA