Dyneon™

Fluorothermoplastics

dyneon

A 3M Company

Injection Molding Guide for Dyneon™ PFA

Table of Contents

Injection Molding for Dyneon™ PFA

INTRODUCTION

Dyneon™ PFA is a high performance melt-processable fluorothermoplastic made from tetrafluoroethylene and perfluorovinylether. Dyneon PFA is characterized by exceptional heat and chemical resistance, superior weather resistance and excellent electrical properties.

Dyneon's 6500 series of PFA is available in different melt flows to fit specific applications and processes. Dyneon PFA is also available as ultra high purity (UHP) material for semiconductor and other critical high purity applications.

Features of Dyneon PFA include:
• Wide service temperature range from -200° to +260°C (-328°F to 500°F)
• Exceptional chemical resistance
• Outstanding mechanical properties
• Very good non-stick properties
• Superior electrical properties
• Non-flammable / no flash point under normal conditions
• Exceptional resistance to environmental stresses such as weathering and aging

PRODUCT PROPERTIES

Table 1 lists material properties that are useful in injection molding applications.

Table 1 Material Properties

Property	**Units**	**Test Method**	**6502*** N, UHP	**6505** N, UHP	**6510** N	**6515** N, UHP
Specific Gravity	g/cm^3	ASTM D792/ ISO 12086	2.15	2.15	2.15	2.15
Melting Temperature	°C (°F)	ASTM D3418/ ISO 12086	308(585)	308(585)	308(585)	308(585)
Melt Index 372/5	g/10min	ASTM D3307/ ISO 12086	2.1	4.5	10	15
Tensile Strength (at break)	MPa (kpsi)	ASTM D 3307/ ISO 12086	30(4.3)	30(4.3)	27(3.9)	26(3.8)
Elongation	%	ASTM D 3307/ ISO 12086	380	410	450	450
Flexural Modulus	MPa (kpsi)	ASTM D 790/ ISO 12086	550(80)	550(80)	600(87)	620(90)

* 6502 is not generally recommended for injection molding operations because of its high melt viscosity.
N: Standard applications
UHP: Ultra High Purity

EQUIPMENT SPECIFICATION

Clamp Tonnage

The clamp tonnage requirements for PFA are generally lower than for most polymers. PFA has a high melt viscosity that leads to large pressure drops across the gate and runner system and injection velocities must be reduced so that pressures can be kept low to prevent melt fracture as well as part fracture. Therefore, clamp tonnage requirements for PFA are generally low. The high viscosity of PFA does however reduce the occurrence of part flashing. For most applications, 0.5 tons per cm^2 (3 tons per in^2) of projected part and runner area is sufficient.

Injection Unit

Injection unit pressure requirements are low for PFA. High injection velocities and pressures are undesirable because they cause melt fracture and part fracture. It is recommended to keep injection pressures less than 1000 bars (15,000 psi) and pack pressures less than 700 bars (10,000 psi).

Barrel and Screw

Barrel Sizing

The barrel should be sized to contain between 1.2 to 4 times the shot capacity. If the shot capacity is greater than 4 times the shot size, the increased residence time can cause thermal degradation of the PFA.

Materials of Construction

PFA is corrosive to common steel at usual melt processing temperatures. All parts of the machine that contact molten PFA and its vapors must be made using corrosion resistant metals. Table 2 lists commonly used screw and barrel materials for processing PFA.

Table 2 Common Barrel and Screw Materials for Dyneon PFA

Barrel Material	Screw Materials
Xaloy® 309[1]	Hastelloy® C276[2]
IDM® 260[3]	Inconel® 625[4]
Wexco® B022[5]	Haynes® 242™ alloy[2]
Reiloy® 115[6]	Inconel® 725[4]

Screw Design

Figure 1 depicts the screw characteristics that are most important to consider in injection molding operations. Table 3 provides recommended ratio ranges for each screw characteristic as it relates to the screw diameter. To obtain the dimension of each characteristic, multiply the diameter of the screw by the screw diameter ratio.

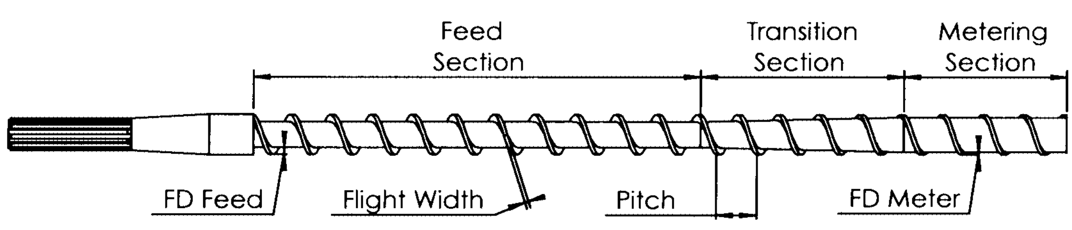

Figure 1 Screw Diagram

Table 3 Typical Screw Characteristics

Characteristic	Description	Screw Diameter Ratio Multiply this number by the Screw Diameter
Length	Length of flighted section of the screw	20
Feed Section	Length of feed section	10 to 12
Transition Section	Length of transition section	4 to 5
Metering Section	Length of metering section	4 to 5
Pitch	Distance between flights	1
Flight Width	Width of flight	0.1
FD Feed	Flight depth in the feed section	0.16 to 0.18
FD Metering	Flight depth in the metering section	0.06 to 0.07
Compression	FD Feed/FD Meter	2.5 to 2.7

Nozzle and Nozzle Tip

The material path in the nozzle and nozzle tip should be smooth without pockets or rough transitions where polymer can stagnate and, over time, degrade. Figure 2 shows a cross section of a typical nozzle / end cap configuration. As shown in Figure 2, the screw tip and the end cap should be drafted with the same angle to prevent a dead spot between the two. The mate between the nozzle tip and the nozzle should be the same internal diameter to avoid a step.

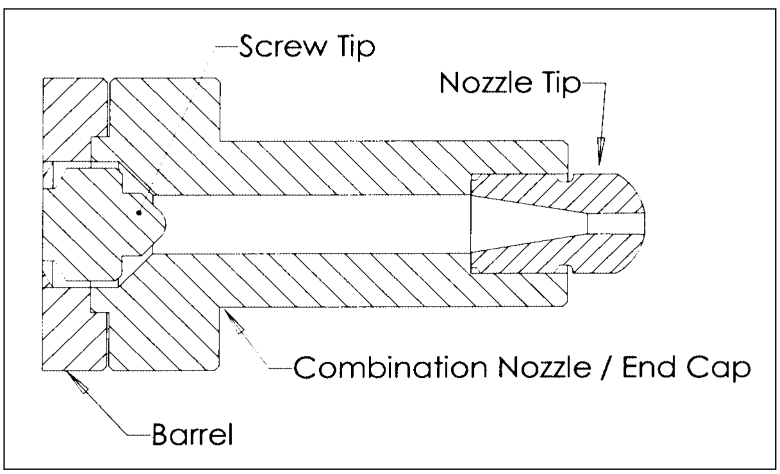

Figure 2 Combination Nozzle and End Cap

Figure 3 shows the details of the nozzle tip. The discharge end of the nozzle tip should be reverse tapered for the first 3 diameters (D1 *3) of the nozzle length with a 1 mm per 50 mm taper (0.25" per foot taper). The discharge hole diameter should be 0.25 - 0.5 mm (0.01"- 0.02") smaller than the matching orifice in the sprue. The diameter of the nozzle side orifice (D2) should match the internal diameter of the nozzle. A common material of construction for the nozzle tip is Hastelloy® 718 alloy since it is harder than most other corrosion resistant materials.

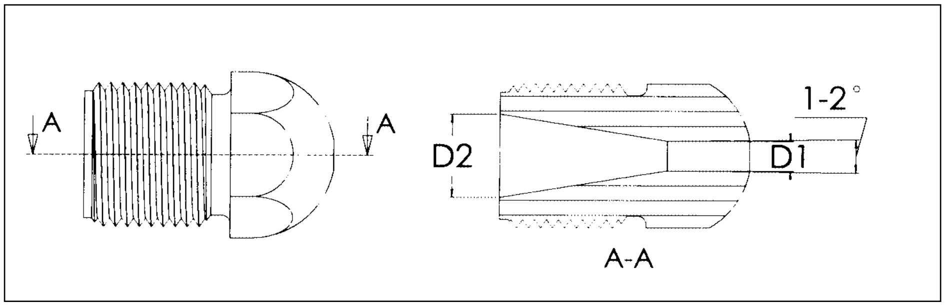

Figure 3 Nozzle Tip

Screw Tip Assemblies

A commonly used screw tip assembly for fluoropolymers is the check ring. Figure 4 shows a locking check ring and screw tip assembly. Corrosion resistant materials are relatively soft and the seats on the check ring and screw tip wear rapidly when the check ring is allowed to float. To prevent this wear the check ring is usually locked. The screw tip should be as smooth as possible. Keep the flats on the screw tips small with only enough flat area to positively secure or remove the screw tip assembly and the screw.

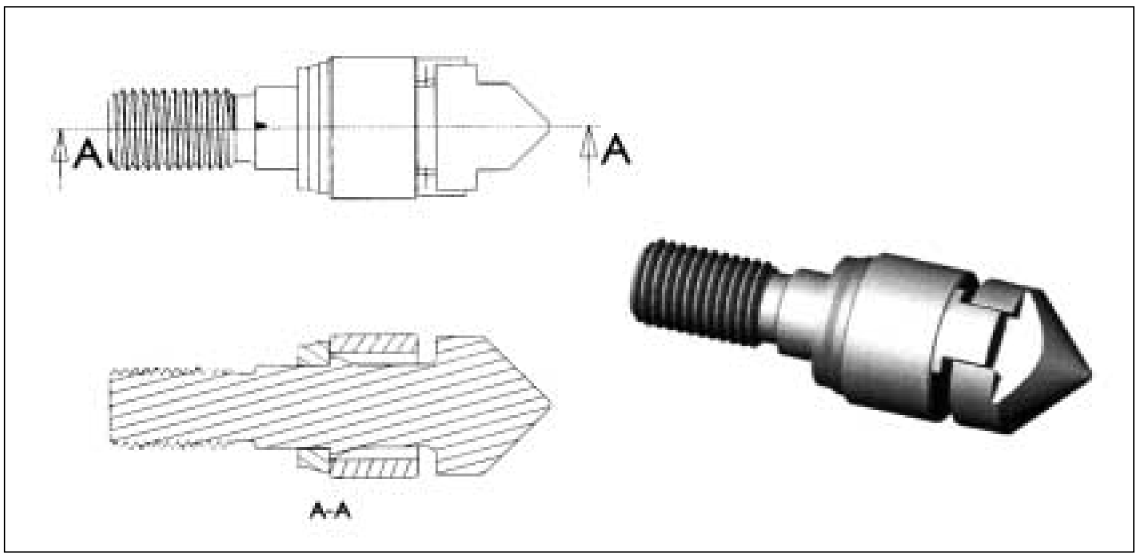

Figure 4 Locking Check Ring Assembly

Mold Materials and Design

Materials of Construction

Any part of the mold that contacts molten PFA must be corrosion protected. Since corrosive gasses are liberated during processing, the vent system should also be protected.

The mold cavity can be constructed from either a nickel alloy or plated tool steel. Nickel alloys are softer, more difficult to machine, and are poorer heat conductors than plated tool steel. Consequently molds are often machined from tool steel and then plated using an electroless nickel plating process. If a higher polish, lower friction, or longer wear is desired, nickel / PTFE coatings such as Polyond™[7] or Nickelon™[8] can be used. As the tool steel material in the core and cavity is protected from corrosion by the plating, the mold surfaces need to be replated before the finish wears through. Replating is also required following modifications or repairs. These are the main disadvantages associated with the use of plated molds.

When plated molds are stripped and re-plated it is important to use a stripping process that will not attack the base metal. Generally the faster the stripper the more chance there is of attacking the base metal. Thus, it is recommended to use a slower alkaline stripping process to avoid attack of the base metal.

Runners and Gates

Proper design of the gate and runner system is important for PFA. PFA is a high melt viscosity material and will melt fracture easily if sharp comers are present or if the runner or gate cross section is too small. It is advisable to keep the runner and gate system large, smooth, streamlined, well radiused and as short as possible. Should the runners or gates be too small, the part may not fill properly or it may need to be filled so slowly that it will have a poor surface finish.

Fan Gating

The use of fan gates is recommended for many part configurations when injection molding PFA. The use of fan gates result in the superior distribution of the molten material into the mold cavity therefore reducing the occurrence of delamination within the part, and minimizing the differential cavity pressure (maximum cavity pressure - minimum cavity pressure).

A good illustration of a fan gate is given in Figure 5. The width of the gate at the part (W1) should typically range from 20 percent of the part width (W2) to 100 percent of the part width. The fan should be flared into the part with a 1 mm (0.04”) or greater fillet (R2). As part sizes are increased, the fillet (R2) should also be increased. The wider the fan (W1) the easier the part will mold; however, narrower fans are easier to trim and use less material. Figure 5 summarizes the design guidelines for fan gating.

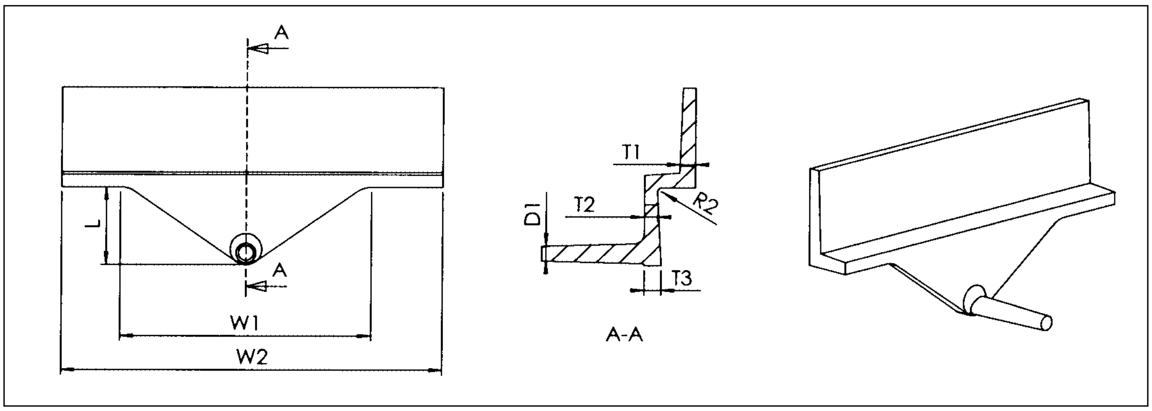

Figure 5 Fan Gating

Table 4 Fan Gating Guidelines

L	1/2 W1
W1	20 - 100% of W2
T2	75 - 100% of T1
D1	1x - 3x part thickness
R2	> 1 mm (0.040”)
R1	> 1 mm (0.040”)
T3	1x - 2x T1 (part thickness)

Sprue Gating

Sprue gating is commonly used in single cavity tools. Figure 6 shows a radially balanced part gated at the center using a sprue gate. It is generally necessary to have the sprue diameter larger than the part thickness to reduce melt fracture and delamination. In the case of larger parts the sprue diameter can be as much as 4 times the part thickness. As with all gating methods the sprue should be flared into the part with a generous radius. Sprue gate guidelines are summarized in Table 5.

Table 5 Sprue Gate Guidelines

A1	2-3°
R1	> 1.5 mm (0.060")
D1	1.25x - 4x T1 (part thickness)

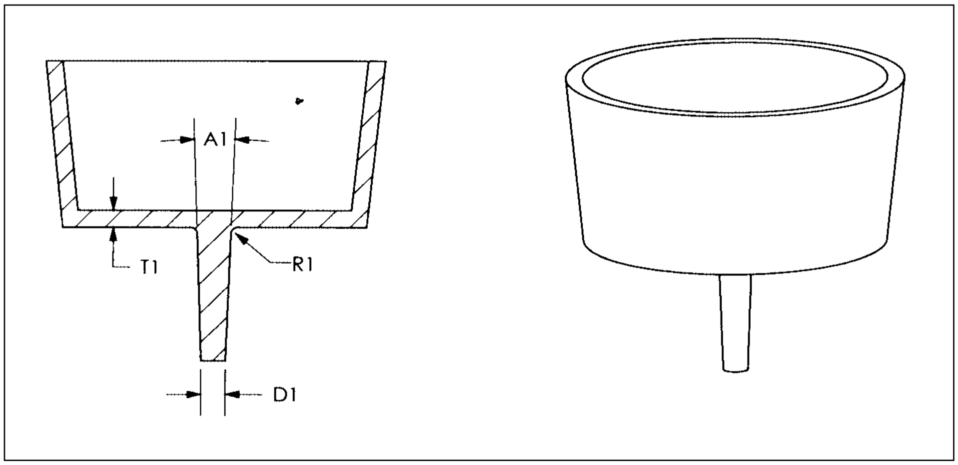

Figure 6 Sprue Gating

Tunnel Gating

Tunnel gates are generally not recommended when molding PFA. The use of small diameter tunnel gates cause higher shear rates which can lead to melt fracture. However, there are instances where their use may be appropriate when the parts are small.

Runner Systems for Multiple Cavities

For multiple cavity molds it is preferable to make the runner system balanced and keep the runners as short as possible. The gate should be between 1/2 x to 1 x part thickness.

Figure 7 shows an example of an unbalanced eight-cavity runner system. Unbalanced runner systems cause the cavities that are closest to the sprue to receive a higher pressure relative to the cavities that are further away. Differences in pressure can cause part dimensions to vary widely. Unbalanced runners can significantly narrow the processing window by making it difficult to fill the parts furthest from the sprue while avoiding melt fracture in the parts closest to the sprue.

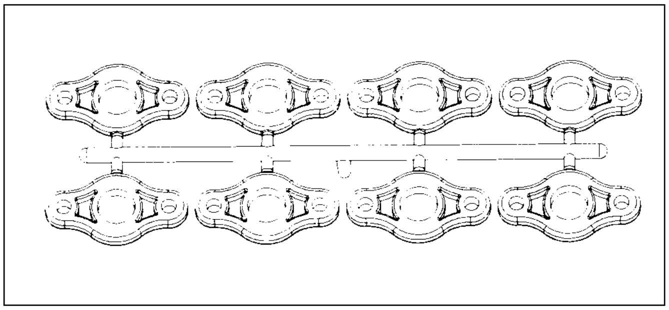

Figure 7 Unbalanced Runner System

Figure 8 shows a part similar to the one depicted in Figure 7 but, in this case, using a balanced runner system. All cavities will have the same cavity pressure and will experience the same injection velocities. A balanced runner system is recommended as it allows for a wider processing window and a high level of consistency.

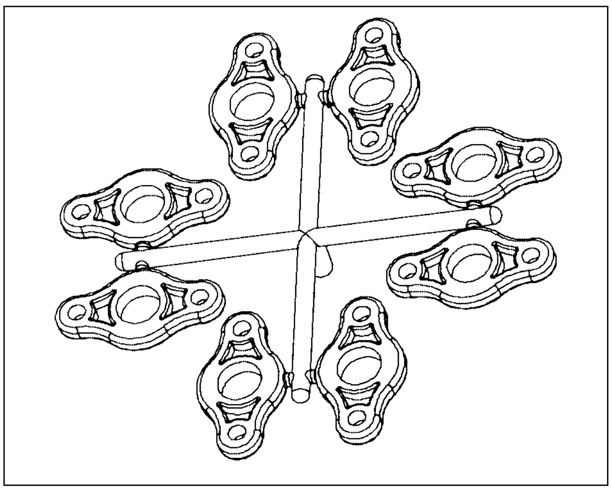

Figure 8 Balanced Runner System

Hot Runner Systems

The weight of the cold runner system on small injection molded parts is often greater than the weight of the part. A cost analysis, balancing the cost of a hot runner mold against the cost of scrap from the cold runner system, can in some cases show a hot runner system to be cost effective. Hot runner systems can be used with PFA provided the hot runner system is constructed from highly corrosion-protected materials. For instance an Inconel 625 hot runner manifold along with a molybdenum nozzle can be used with good results[9].

The hot runner channels should be streamlined using generous radii on all bends and corners. The length of the runner system should be as short as possible. An open hot runner nozzle is recommended. If a valve gate is necessary, use a valve system that retracts amply to prevent excessive shear to the material. Choose a design without sharp corners or steps to avoid dead zones and high shear. Use a short cold sprue and / or a fan to minimize part flatness problems that can be caused by higher mold temperatures near the gate.

Figure 9 shows an example of a hot nozzle going into a short sprue. Figure 10 shows the placement of a hot nozzle going into a cold fan in a single cavity configuration. Figure 11 shows hot runners going into a part with multiple fan gates. The hot runners are shown in solid lines and the cold portion of the system, the part and the fan gates, are shown in dashed lines.

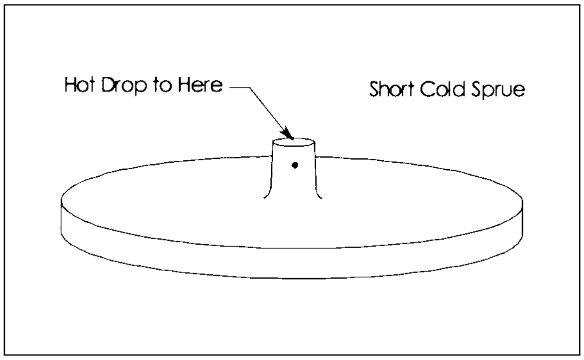

Figure 9 Hot Runner into a Short Cold Sprue

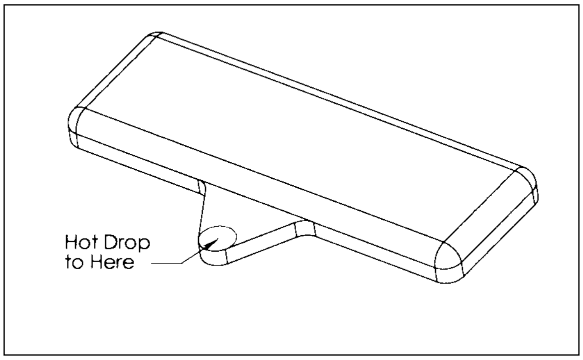

Figure 10 Hot Runner into a Cold Fan Gate

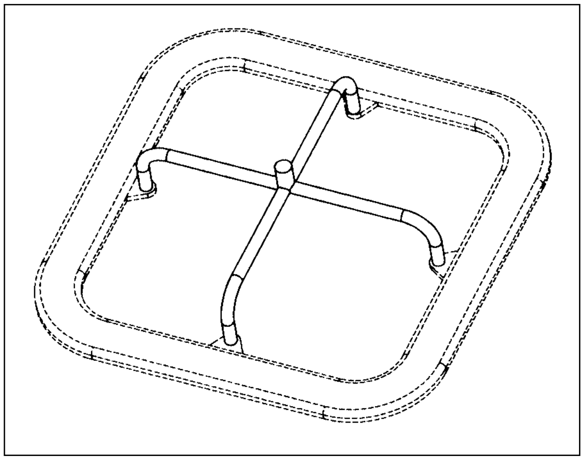

Figure 11 Hot Runner into Fan Gates

Vents

Vents should be placed in the cavity and along the runner system wherever gasses can be trapped. The most important locations for vents are the cold slug traps, knit lines and the last areas of the mold to be filled with polymer. Vent depths can be deeper with PFA compared to most plastics because of the high melt viscosity of PFA. It is possible to use vent depths of 0.1 mm (0.004") without flashing. However, it is recommended to make the vent depth between 0.05 - 0.075mm (0.002" – 0.003"). If the vent areas are well polished they will have a tendency to remain clean.

PART SHRINKAGE

Part shrinkage is primarily dictated by the polymer. However, shrinkage is also influenced by equipment design and processing conditions. These include runner and gate design, flow direction, part thickness, mold temperature, cavity pressure, melt temperature and cooling time. For this reason, it is recommended to leave dimensionally critical areas of the mold "steel safe" when developing new parts. This can be achieved by following the simple steps given below:

- leave extra mold steel in areas that have strict dimensional requirements
- mold the part using an ideal process
- determine the part shrinkage in each of the dimensionally critical areas
- cut the steel to final dimensions using the determined shrinkage from the previous step
- ensure the parts are dimensionally correct
- polish the core and cavity
- plate the mold

The "steel safe" method is best used for width and length dimensions and not for thickness since changing the thickness will change the shrink value.

Table 6 shows typical shrinkage values obtained from molding a 4 inch diameter disk gated from the end.

Table 6 Typical Mold Shrinkage Values for PFA

Part Thickness	1.59mm (0.0625")	3.18mm (0.125")	4.76mm (0.1875")	6.35mm (0.25")
% Shrinkage Parallel to Flow	4.0-4.5	4.4-4.9	5.2-5.7	5.5-6.0
% Shrinkage Perpendicular to Flow	4.3-4.8	4.5-5.0	5.3-5.8	5.5-6.0

Listed below are several issues to consider in addition to the data provided in Table 6.

- Cavity pressure is the processing condition that has the most significant impact on part shrinkage. Increasing the cavity pressure will decrease shrinkage. Higher cavity pressures can be obtained by shortening the runner system, enlarging runner and gate diameters and increasing the pack and hold pressures on the machine.
- PFA, like other polymers, can retain molecular orientation in the direction of flow. The material will shrink less in the direction of flow and more in the direction perpendicular to the flow. As the part cross section becomes thinner this effect becomes more prominent.
- Slowing the cooling rate can cause the part to shrink more since the material has more time to crystallize. Slower cooling rates will also allow more relaxation of internal stresses. Slower cooling rates occur with increased part thickness and higher mold temperatures.
- In many molds, some or all of the part dimensions are constrained from shrinking by steps, bosses, and / or cores. Those regions in the mold that are constrained are referred to as fixtured regions. As constrained areas are cooled more in the mold, less shrinkage will be observed in these fixtured areas. Increased cooling in the mold can be caused by decreased mold temperatures, decreased part thickness and increased cooling time. Constraining certain regions of the mold from shrinking results in localized internal stress. When the part is subjected to heat in service, these stresses will begin to relax, which can cause the part dimensions to change. Part warpage can occur where there is a combination of fixtured and unfixtured regions in a part resulting in uneven shrinkage.

ANNEALING

Annealing, or heat treating, is used to reduce internal stresses, increase part crystallinity and reduce the probability that the parts will warp or change dimensions while in service. Annealing may also be used in conjunction with fixtures to straighten parts that are warped because of a requirement for an uneven wall thickness or uneven cooling. Annealing is utilized when parts require stable dimensions, are intended for use at elevated temperatures or need maximum physical properties.

Annealing of PFA is accomplished by

- heating the parts up to between 240°C and 270°C (460°F and 520°F)
- holding the oven at annealing temperature for a specified time
- allowing the parts to cool gradually by either a controlled ramp or by turning the oven off and allowing it to slowly cool

In many instances it is desirable to keep annealing temperatures uniform so dissimilar parts can be annealed in the same batch and oven temperatures never need to be adjusted. A recommended standard annealing temperature is 250°C (480°F). The recommended soak time is 10 minutes per millimeter (0.04") of part thickness. If the oven has a controlled ramp, the oven should be ramped down to room temperature at 10 minutes per millimeter of part thickness - the same length of time as the soak period. Table 7 shows typical values for annealing shrinkage at various part thickness.

Table 7 Annealing Shrinkage

Part Thickness	1.59mm (0.0625")	3.18mm (0.125")	4.76mm (0.1875")	6.35mm (0.25")
Time (Minutes) at 250°C (482°F)	20	30	45	60
% Shrinkage Parallel to Flow	0.4-0.6	0.3-0.5	0.1-0.3	0.1-0.3
% Shrinkage Perpendicular to Flow	0.6-0.7	0.4-0.5	0.2-0.4	0.1-0.3

TYPICAL PROCESSING CONDITIONS

Barrel Temperature

The relationship between the screw and barrel shot capacity and the shot size or cavity volume will dictate the material's residence time in the barrel and, consequently, the temperature settings. For instance, if the shot size is small relative to the shot capacity of the machine, the residence time will be long and it will be necessary to lower the barrel temperatures. If the shot size requires almost all of the barrel capacity and the residence time is relatively short, then the temperatures will need to be higher. Also, filling thin cross sections generally requires higher melt temperatures to reduce the viscosity. Table 8 shows recommended temperatures for the barrel and the mold. A recommended starting profile for the barrel temperatures is 360°C (680°F) for the rear zone, 365°C (690°F) for the center zone and 370°C (700°F) for the front zone and the nozzle.

Table 8 Barrel and Mold Temperature Setpoints

Zone	Temperature °C / °F
Rear	350-380 / 660-715
Center	350-390 / 660-735
Front	360-400 / 680-750
Nozzle	360-400 / 680-750
Melt	360-400 / 680-750
Mold	180-260 / 356-500

Viscosity loss is an indicator that PFA has undergone degradation with consequent property loss. A measure of degradation from both shear and time at temperature can be obtained by making parts from material that has been held in the injection molder's barrel for various times and temperatures. Once these parts are molded the melt flow index (MFI) can then be measured.

Figure 12 is a contour graph showing the effect of residence time at various temperatures on the MFI of Dyneon™ PFA 6515N. The region indicated with a "**+**" sign shows the desirable process window for the material. The regions indicated with a "**–**" sign show a MFI shift in excess of 20%. The material will degrade at a defined rate at a given temperature and the rate at which degradation takes place increases exponentially with temperature.

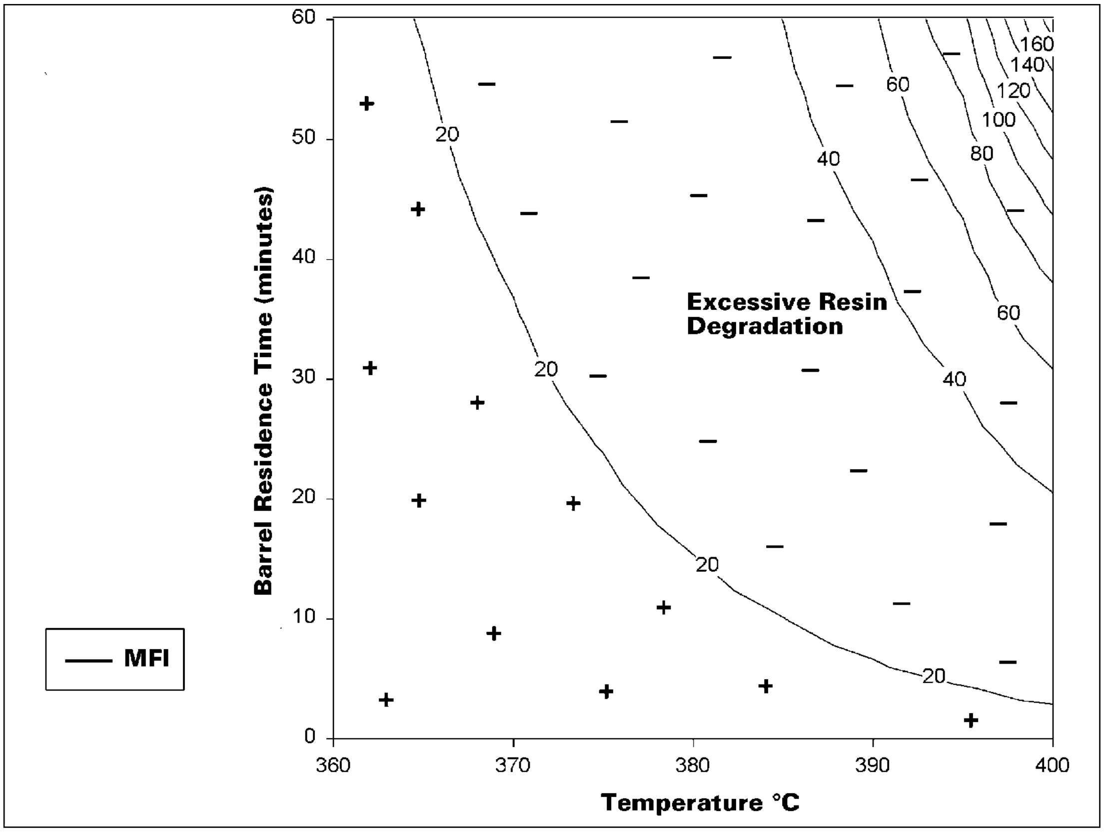

Figure 12 Desirable Times and Temperatures

Mold Temperature

Determination of the mold temperature setpoint should consider part geometry, desired surface finish, stress / warpage, delamination, ejection, shrink and cycle time. Higher mold temperatures will assist in the fill of thin wall sections, will reduce part stress, improve part crystallinity, and, in some applications, may eliminate the need for annealing the part as a secondary operation. However, higher temperatures lengthen cycle times and may increase part warpage. Mold temperatures should range from 180°C (356°F) to 260°C (500°F). A recommended starting temperature is 200°C (400°F).

Injection Velocity

Polymer resins will melt fracture when the applied stresses exceed the melt strength of the resin. The velocity at which melt fracture occurs is called the critical velocity. The critical velocity for PFA is lower than for most other thermoplastic materials. If a frosty appearance is seen on the surface of the part around the gate, by corners and other abrupt transitions, it is likely that the critical velocity has been exceeded.

Injection velocities that are too slow will result in very small (almost microscopic) pits in the surface of the parts. Slow filling causes these pits as the PFA solidifies against the surface of the mold without enough pressure to pack it against the mold wall. Extremely slow inject velocities will leave a rough surface and can result in a short shot.

Typical injection velocities for PFA are between 3 and 15 mm / sec (0.1 to 0.6 inches / sec). It is best to start the inject at 3 to 4 mm / sec and incrementally ramp the velocity up to 10 - 15 mm / sec, then ramp the velocity down the last 5 - 10% of fill. For the final 5 - 10% of fill, choose a velocity so the fill pressure at the end of fill is close to the desired pack pressure. This will result in a smooth and controlled transfer to pack.

Shot Size and Cushion

To determine the shot size and cushion, turn off the pack and hold stages by setting the timers to zero. Set the shot size to partially fill the part and increase the shot size after each shot until the part is 98% full. Once the parts are 98% full, set the pack time and pressure. Increase or decrease both the shot size and the velocity to pressure transfer (VPT) point in equal amounts until the cushion is between 4 mm (0.15") and 7 mm (0.25"). If injection velocities, pack pressures, pack times or material temperatures are changed, this process should be repeated.

Pack Pressure and Pack Time

Pack pressures must be kept low to prevent the fracturing of solidified resin in and around the gate area or in other restricted areas in the part. On closed loop machines when many stages of pack or hold are available it can be helpful to ramp the pack pressure down over time. Pack pressures over 550 bar (8000 psi) are rarely necessary. A recommended strategy is to pack the part for 2 - 5 seconds at a 375 - 475 bar (5000 - 7000 psi) and then lower the pack pressure to 250 - 350 bar (3500 - 5000 psi) for the remainder of the pack / hold time. If the sprue or runner system appears fractured, the pressure has been too high for too long. If the part has excessive sink marks, pack pressures or times should be increased.

Back Pressure / Screw Recovery Speed

Back pressure is the pressure the screw must generate before the screw can recover. By increasing the back pressure, more of the heat put into the material is generated by shear, and both material mixing and color dispersion improve. High back pressures however, can also degrade resin, and cause the bulk temperature of the polymer to exceed the temperature setpoint. It is usually desirable to keep the back pressure as low as possible while obtaining sufficient mixing of the material. A recommended starting point for back pressure is 50 - 70 bar (750 psi - 1000 psi) of specific pressure.

Because PFA has a much higher heat capacity than most thermoplastic materials and the gate and runner system is relatively large, it may be impractical to pack and hold the part until the gate or sprue is completely frozen. When screw recovery starts before the gate freezes off, back pressure will act as an additional pack or hold stage. As the back pressure is set higher, more material will be packed onto the part.

Screw recovery speed should be kept as slow as possible. Use approximately 90% of the cooling time for screw recovery. A fast recovery speed has three drawbacks. First, with a quick recovery speed, ultra-high shear is generated, creating localized hot spots in the melt that can far exceed the desired material temperature – this can lead to resin degradation. Second, the screw resides in its full back position for most of the cycle with no agitation. This results in a stagnant melt next to the barrel wall which becomes a different temperature than the material in the center of barrel, creating a non-uniform melt. Finally, rapid screw recovery can lead to faster wear of the screw and barrel. There are two sources of heat in the melt - shear and heater bands. Higher back pressure and higher screw speeds will cause a higher proportion of the heat to be generated from shear. As more heat is generated by shear, there is more friction between the polymer and the screw and barrel resulting in faster screw and barrel wear. If better mixing is required, it is best to incrementally increase the back pressure while leaving the screw speed slow.

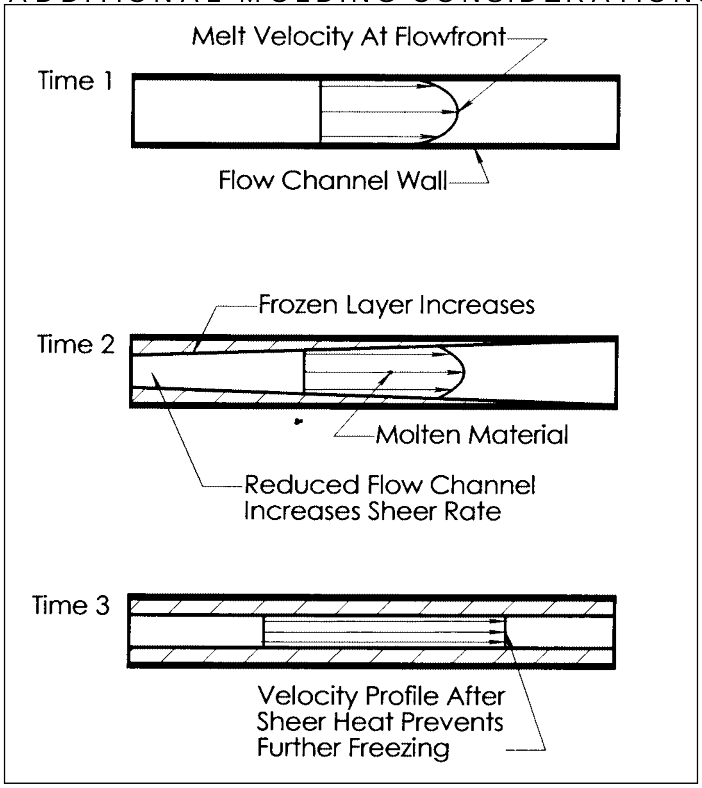

Figure 13 Delamination Mechanism

Delamination

PFA parts have a tendency to delaminate in a radial pattern around the gate or sprue. Figure 13 illustrates the mechanism by which delamination can occur during injection molding of PFA.

Time 1 in Figure 13 shows the velocity profile near the flow front. During fill the material begins to freeze against the mold wall as seen in Figure 13 Time 2. As the frozen layer of material increases in thickness, the narrowing flow channel causes increased resistance to material flow. Resistance to flow causes more heat to be generated by shear. At some point during the fill the shear heat being generated becomes sufficient to prevent further freezing of material against the mold wall. When this condition, shown in Figure 13 Time 3, is reached, a boundary is formed between the frozen material and the molten material. The material inside this boundary layer moves with a plug flow through the flow channel with little or no molecular chain entanglements with the molecules in the frozen layer, resulting in laminated layers.

Delamination in the part occurs predominantly around the gate or sprue because a great deal of material has to pass through the flow channel creating the loss of molecular entanglements between layers (as described in the previous paragraph). With higher injection velocities, the frozen layer becomes thinner since more shear heat is being generated. If high injection velocities cause the boundary layer to become too thin, a layer of PFA may peel off from around the gate of a molded part. The best solution for this problem is to flare the runner into the part as much as possible. It can also be helpful to increase the mold temperature and decrease the injection velocity. Lower injection velocities along with higher mold temperatures reduces the chances of forming an unentangled boundary between the frozen layer and the molten material.

Safety

Common safety practices such as wearing safety glasses, not disabling processing equipment safety features and keeping the work area clean are to be followed under all circumstances. Along with these standard safety practices it is important to have good ventilation and thermal protection. For a complete treatment of safe handling practices please refer to either of the following publications:

Guide to the Safe Handling of Fluoropolymer Resins[10]
Guide for the Safe Handling of Fluoropolymer Resins[11].

Ventilation

A ventilation hood should be used to remove gasses that are emitted by molten PFA. It is also desirable to have proper ventilation over all areas where hot parts are placed after being removed from the mold.

Thermal protection

During production, the mold, the material and the newly formed parts are extremely hot. Wear gloves, arm protection and long pants. Never disengage the purge guard. Always wear eye protection.

TROUBLE SHOOTING

Problem	Solution
Frosty appearance around the gate or sprue caused by melt fracture	Decrease the injection velocity Increase the mold temperature Increase the material temperature Increase the diameter of gate or sprue Ensure that all surfaces and transitions are streamlined Add generous radii to the sprue or gate
Delamination around the gate	Decrease the injection velocity Increase the mold temperature Lower pack pressure Increase the diameter of the runner / gate / sprue
Excessive sink marks	Increase the pack pressure Increase the pack time Design part with common wall thicknesses Increase the diameter of the runner / gate / sprue Decrease the thickness of the rib or boss
Part warpage	Increase cool time Decrease mold temperature Design part with common wall thickness Redesign the cooling system to cool evenly
Weak weld lines	Increase mold temperature Increase material temperature Increase inject velocity Improve venting along the weld line
Discoloration of material	Ensure every part of the barrel assembly is corrosion protected Use a smaller barrel assembly if the barrel contains more than 4 shots of material Eliminate dead spots in the barrel, screw tip or nozzle assembly Reduce the shot cushion volume Check for contamination from other materials (e.g. mold release) Verify if material was left in the barrel for an excessive amount of time Reduce melt temperature
Parts too small	Remove restrictions in the material flow path by opening up the nozzle, sprue, runners and gates Increase pack pressure Increase pack time Ensure the cushion is being held - if not, increase shot size and VPT* by equal amounts and make sure the check ring is sealing
Parts too large Ejector pin marks	Decrease pack pressure Increase the size or number of ejector pins Increase draft on the part Increase cool time Check the mold for undercuts Decrease ejection speed
Flash	Decrease the pack pressure Ensure the VPT* occurs before the part is full Ensure the mold is shutting off properly

*VPT = velocity to pressure transfer

TROUBLE SHOOTING (continued)

Problem	Solution
Jetting	Reduce initial injection velocity Increase the size of the gate Redirect the gate to obtain initial flow impingement against a mold wall (usually by gating perpendicular to the mold wall) Add a more generous flare or radius on the part side of the gate
Short shot	Remove restrictions in the material flow path by opening up the nozzle, sprue, runners and gates Increase pack pressure and / or time Increase mold temperature Increase material temperature Ensure the cushion is being held - if not, increase shot size and VPT by equal amounts and make sure check ring is sealing Increase injection velocity Make sure the VPT is set at 98% part fill
Flow from nozzle Parts sticking in the mold	Increase decompression distance Decrease pack pressure Increase draft on the part Plate the mold with PTFE impregnated nickle plating
Bubbles in the part (bubbles will expand when a freshly molded part is heated to near melt temperature)	Lower barrel temperatures Use a barrel capacity of less than 4 times the shot size Use a screw with a higher compression ratio
Shrink voids / vacuum voids (vacuum voids look like a bubble but will collapse when a freshly molded part is heated to near melt temperature)	Design part with common wall thickness Increase pack pressure Increase mold temperatures Verify a consistent cushion Remove restrictions in the material flow path by opening up the nozzle, sprue, runners and gates Make sure the fill is from thick to thin

[1]Xaloy
102 Xaloy Way,
Pulaski, Virginia 24301

[2]Haynes International
1020 W. Park Avenue
P.O. Box 9013
Kokomo, Indiana

[3]IDM-Inductamatals Corporation
101 West Grand Avenue Suite 504
Chicago, IL 60610

[4]Inco Alloys International, Inc.
3200 Riverside Drive
Huntington, WV 25705-1771

[5]Wexco Corporation
1015 Dillard Drive P.O. Box 4297
Lynchburg, VA 24502

[6]Reiloy Metall GmbH
Spicher Straße
D-53829 Troisdorf

[7]Poly Plating Co, Inc.
2096-T Westover Road
Chicopee, MA 01022 USA

[8]Bales Mold Service
2824 Hitchcock Avenue
Downers Grove, IL 60515

[9]Thermojert, PSG Plastic Service GmbH,
68309 Mannheim, Germany

[10]Guide to the Safe Handling of Fluoropolymer Resins
Catalog No. BP- 101
The Society of Plastics Industry, Inc.
1801 K Street, NW, Suite 600K
Washington, DC 20006-1301
202 974 5200

[11]Guide for the Safe Handling of Fluoropolymer Resins
Association of Plastics Manufacturers in Europe
Avenue E. Van Nieuwenhuyse 4
Box 3 B-1160 Brussels
Tel. 32 2 675 3279
Telefax 32 2 675 3935

A 3M Company

Technical Information and Test Data

Technical information, test data, and advice provided by Dyneon personnel are based on information and tests we believe are reliable and are intended for persons with knowledge and technical skills sufficient to analyze test types and conditions, and to handle and use raw polymers and related compounding ingredients. No license under any Dyneon or third party intellectual rights is granted or implied by virtue of this information.

Important Notice:

Because conditions of product use are outside Dyneon's control and vary widely, user must evaluate and determine whether a Dyneon product will be suitable for user's intended application before using it. **The following is made in lieu of all express and implied warranties (including warranties of merchantability and fitness for a particular purpose): If a Dyneon product is proved to be defective, Dyneon's only obligation, and user's only remedy, will be, at Dyneon's option, to replace the quantity of product shown to be defective when user received it or to refund user's purchase price. In no event will Dyneon be liable for any direct, indirect, special, incidental, or consequential loss or damage, regardless of legal theory, such as breach of warranty or contract, negligence, or strict liability.**

Issued: 3/01

Printed in USA
98-0504-0988-1

Dyneon Sales Offices:

Dyneon LLC
6744 33rd Street North
Oakdale, MN 55128
Phone:+1 800 723 9127
+1 651 733 5353
Fax:+1 651 737 7686

Dyneon GmbH & Co. KG
Werk Kelsterbach, Gebäude R279
D-65444 Kelsterbach
Germany
Phone: +49 6107 772 516
Fax: +49 6107 772 517

Dyneon GmbH & Co. KG
Werk Gendorf
D-84504 Burgkirchen
Germany
Phone: +49 8679 7 4709
Fax: +49 8679 3992

Dyneon LLC
50 Milton Drive
Aston, PA 19014-2293, USA
Phone: +1 610 497 8899
Fax: +1 610 497 7050
Toll Free: +1 800 554 6782

Product Information:

+ 1 651 733 5353
+ 1 800 723 9127

www.dyneon.com