3M™ Dyneon™
Fluoroplastics

Injection Moulding Handbook
Handbuch Spritzguss

3M
6 Typical Processing Conditions  27
  6.1 Barrel Temperature  27
  6.2 Mould Temperature  29
  6.3 Injection Velocity  29
  6.4 Shot Size and Cushion  30
  6.5 Pack Pressure and Pack Time  30
  6.6 Back Pressure / Screw Recovery Speed  30
7 Additional Moulding Considerations  32
  7.1 Delamination  32
  7.2 Safety  34
8 Trouble Shooting  35
9 Appendix  39

<< First part of table of content on page 2
1. Introduction

3M™ Dyneon™ Fluorothermoplastics are high performance melt-processable thermoplastic materials made from tetrafluoroethylene and other fluorinated and non-fluorinated monomers. Dyneon Fluorothermoplastics are characterized by exceptional heat and chemical resistance, superior weather resistance and excellent electrical properties.

3M™ Dyneon™ Fluoroplastic PFA 6500 series 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 PFA materials include:

- Wide service temperature range from -200 °C 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
3M™ Dyneon™ Fluoroplastic FEP 6300 series is available in different melt flows to fit specific applications and processes.

Features of FEP materials include:

- Excellent dielectric properties
- High thermal stability
- Service temperature rating up to more than 200 °C
- Outstanding chemical resistance
- Wide processing window
- Greatly enhanced flex life properties
- Extremely low flammability (high LOI)
- Smooth surface
- Excellent anti-stick properties
- Low coefficient of friction
- Very high weathering and UV stability
- Improved mechanical properties
3M™ Dyneon™ Fluoroplastic ETFE 6200 series is available in two different melt flows to fit specific applications and processes.

Features of ETFE materials include:

- Extremely high weathering and UV stability
- Wide service temperature range (-200 to +150 °C)
- Excellent tear and tear propagation resistance
- Very good non-stick characteristics
- Good resistance to high energy radiation
- High light transmission in the visible and UV-A range
- Low flammability
- Excellent mechanical properties
- Very good resistance to solvents and chemicals
- Low permeability
2. Product Properties

Table 1 lists material properties of 3M™ Dyneon™ Fluorothermoplastics that are commonly used in injection moulding applications.

<table>
<thead>
<tr>
<th>Property</th>
<th>Unit</th>
<th>Test Method</th>
<th>PFA 6505TZ</th>
<th>PFA 6515TZ /NZ/UHPZ</th>
<th>PFA 6525TZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific Gravity</td>
<td>g/cm³</td>
<td>DIN EN ISO 12086</td>
<td>2.15</td>
<td>2.15</td>
<td>2.15</td>
</tr>
<tr>
<td>Melting Temperature</td>
<td>°C (°F)</td>
<td>DIN EN ISO 12086</td>
<td>310 (590)</td>
<td>310 (590)</td>
<td>310 (590)</td>
</tr>
<tr>
<td>Melt Index</td>
<td>g/10 min</td>
<td>DIN EN ISO 1133</td>
<td>5</td>
<td>15</td>
<td>25</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>MPa (PSI)</td>
<td>DIN EN ISO 527-1</td>
<td>30 (4350)</td>
<td>26 (3770)</td>
<td>20 (2900)</td>
</tr>
<tr>
<td>Elongation at break</td>
<td>%</td>
<td>DIN EN ISO 527-1</td>
<td>410</td>
<td>450</td>
<td>350</td>
</tr>
<tr>
<td>Tensile Modulus</td>
<td>MPa (kPSI)</td>
<td>DIN EN ISO 527-1</td>
<td>550 (80)</td>
<td>620 (90)</td>
<td>630 (91)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>FEP 6307Z</th>
<th>FEP 6322Z</th>
<th>FEP 6338Z</th>
<th>ET 6235Z</th>
<th>ET 6218Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific Gravity</td>
<td>2.14</td>
<td>2.14</td>
<td>2.14</td>
<td>1.72</td>
<td>1.72</td>
</tr>
<tr>
<td>Melting Temperature</td>
<td>258 (496)</td>
<td>252 (486)</td>
<td>250 (482)</td>
<td>266 (511)</td>
<td>266 (511)</td>
</tr>
<tr>
<td>Melt Index</td>
<td>7</td>
<td>22</td>
<td>38</td>
<td>10</td>
<td>18</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>25 (3625)</td>
<td>20 (2900)</td>
<td>20 (2900)</td>
<td>50 (7250)</td>
<td>40 (5800)</td>
</tr>
<tr>
<td>Elongation at break</td>
<td>350</td>
<td>300</td>
<td>300</td>
<td>450</td>
<td>400</td>
</tr>
<tr>
<td>Tensile Modulus</td>
<td>630 (91)</td>
<td>670 (97)</td>
<td>680 (99)</td>
<td>1200 (174)</td>
<td>1100 (160)</td>
</tr>
</tbody>
</table>

Table 1: Material Properties
3. Equipment Specification

3.1 Clamp Tonnage

The clamp tonnage requirements for Fluorothermoplastics are generally lower than for most polymers. They are shear sensitive and injection velocities must be kept low to prevent melt fracture as well as part fracture. Therefore, clamp tonnage requirements for Fluorothermoplastics are generally low. For most applications, about 0.5 tons per cm² (3 tons per sqin) of projected part and runner area is sufficient.

3.2 Injection Unit

Injection unit pressure requirements are low for Fluorothermoplastics. 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).

3.2.1 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 Fluorothermoplastic.

3.2.2 Materials of Construction

Fluorothermoplastics are corrosive to common steel at usual melt processing temperatures. All parts of the machine that contact Fluorothermoplastic melt and its vapors must be made from corrosion resistant metals.
Table 2 shows an overview of commonly used screw and barrel materials for processing Fluorothermoplastics. Ultra high purity applications in particular require the usage of alloys with low iron-content, such as Haynes® 242™, Hastelloy® C-276 or Inconel® 625. These materials are also broadly used in other applications throughout the industry. Internal barrel coatings made of Reiloy® 115 and screws made from Inconel® 625 or Hastelloy® C-276 are well proven.

<table>
<thead>
<tr>
<th>Barrel Materials</th>
<th>Screw Materials</th>
</tr>
</thead>
<tbody>
<tr>
<td>Xaloy® 309&lt;sup&gt;6&lt;/sup&gt;</td>
<td>Hastelloy® C-276&lt;sup&gt;6&lt;/sup&gt;</td>
</tr>
<tr>
<td>IDM® 260&lt;sup&gt;2&lt;/sup&gt;</td>
<td>Inconel® 625&lt;sup&gt;3&lt;/sup&gt;</td>
</tr>
<tr>
<td>Wexco® B0-22&lt;sup&gt;4&lt;/sup&gt;</td>
<td>Haynes® 242™&lt;sup&gt;5&lt;/sup&gt;</td>
</tr>
<tr>
<td>Reiloy® 115&lt;sup&gt;1&lt;/sup&gt;</td>
<td>Inconel® 725&lt;sup&gt;3&lt;/sup&gt;</td>
</tr>
</tbody>
</table>

Table 2: Common Barrel and Screw Materials for Dyneon Fluorothermoplastics (See 9 Appendix for company details)

3.2.3 Screw Design

Figure 1 depicts the screw characteristics that are most important to consider in injection moulding operations.

![Figure 1: Screw Diagram](image)

Figure 1: Screw Diagram
Table 3 shows an overview of typical screw characteristics for the processing of Fluorothermoplastics.

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Description</th>
<th>Screw Diameter Ratio (L/D) (multiply this number by the screw diameter)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>Length of flighted section of the screw</td>
<td>20</td>
</tr>
<tr>
<td>Feed Section</td>
<td>Length of feed section</td>
<td>10 to 12</td>
</tr>
<tr>
<td>Transition Section</td>
<td>Length of transition (compression) section</td>
<td>4 to 5</td>
</tr>
<tr>
<td>Metering Section</td>
<td>Length of metering section</td>
<td>4 to 5</td>
</tr>
<tr>
<td>Pitch</td>
<td>Distance between flights</td>
<td>1</td>
</tr>
<tr>
<td>Flight Width</td>
<td>Width of flight</td>
<td>0.1</td>
</tr>
<tr>
<td>FD Feed</td>
<td>Flight depth in the feed section</td>
<td>0.16 to 0.18</td>
</tr>
<tr>
<td>FD Metering</td>
<td>Flight depth in the metering section</td>
<td>0.06 to 0.07</td>
</tr>
<tr>
<td>Compression Ratio</td>
<td>FD Feed / FD Metering</td>
<td>2.5 to 2.7</td>
</tr>
</tbody>
</table>

Table 3 shows an overview of typical screw characteristics for the processing of Fluorothermoplastics.
3.2.4 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.

![Diagram of Combination Nozzle and End Cap](image-url)
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
3.2.5 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
3.3 Mould Materials and Design

3.3.1 Materials of Construction

Any part of the mould that contacts Fluorothermoplastic melt must be corrosion protected. Since corrosive gases are generated during processing, the vent system should also be protected. The mould 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 moulds 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™ or Nicklon™ can be used.

As the tool steel material in the core and cavity is protected from corrosion by the plating, the mould 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 moulds. When plated moulds 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.

3.3.2 Runners and Gates

Proper design of the gate and runner system is important for Fluorothermoplastics, as they 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.
3.3.2.1 Fan Gating

The use of fan gates is recommended for many part configurations when injection moulding Fluorothermoplastics. The use of fan gates result in the superior distribution of the molten material into the mould 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 mould; however, narrower fans are easier to trim and use less material. Figure 5 summarizes the design guidelines for fan gating.

![Fan Gate Guidelines](image)

<table>
<thead>
<tr>
<th>Equipment Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>L</td>
</tr>
<tr>
<td>W1</td>
</tr>
<tr>
<td>T2</td>
</tr>
<tr>
<td>D1</td>
</tr>
<tr>
<td>R</td>
</tr>
<tr>
<td>T3</td>
</tr>
</tbody>
</table>

Table 4: Fan Gate Guidelines
3.3.2.2 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>
<thead>
<tr>
<th>Sprue Gate Guidelines</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
</tr>
<tr>
<td>R1</td>
</tr>
<tr>
<td>D1</td>
</tr>
</tbody>
</table>

Table 5: Sprue Gate Guidelines

Figure 6: Sprue Gating
3.3.2.3 Tunnel Gating

Tunnel gates are generally not recommended when moulding Fluorothermoplastics. 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.

3.3.2.4 Runner Systems for Multiple Cavities

For multiple cavity moulds 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](image)

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](image)
3.3.2.5 Hot Runner Systems

The weight of the cold runner system on small injection moulded parts is often greater than the weight of the part. A cost analysis, balancing the cost of a hot runner mould 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 Fluorothermoplastics 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.

The hot runner channels should be streamlined using generous radiuses 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 mould temperatures near the gate.

3.3.3 Vents

Vents should be placed in the cavity and along the runner system wherever gases can be trapped. The most important locations for vents are the cold slug traps, knit lines and the last areas of the mould to be filled with polymer. Recommended vent depths are between 0.05 - 0.075 mm (0.002” - 0.003”). If the vent areas are well polished they will have a tendency to remain clean.
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.
4. 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, mould temperature, cavity pressure, melt temperature and cooling time. For this reason, it is recommended to leave dimensionally critical areas of the mould “steel safe” when developing new parts. This can be achieved by following the simple steps given below:

- leave extra mould steel in areas that have strict dimensional requirements
- mould 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 mould

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. In Tables 6a and 6b, typical shrinkage values are exemplified for PFA and ETFE.

<table>
<thead>
<tr>
<th>Part Thickness</th>
<th>1.6 mm (0.0625”)</th>
<th>3.2 mm (0.125”)</th>
<th>4.8 mm (0.1875”)</th>
<th>6.4 mm (0.25”)</th>
</tr>
</thead>
<tbody>
<tr>
<td>% Shrinkage Parallel to Flow</td>
<td>4.0-4.5</td>
<td>4.4-4.9</td>
<td>5.2-5.7</td>
<td>5.5-6.0</td>
</tr>
<tr>
<td>% Shrinkage Perpendicular to Flow</td>
<td>4.3-4.8</td>
<td>4.5-5.0</td>
<td>5.3-5.8</td>
<td>5.5-6.0</td>
</tr>
</tbody>
</table>

Table 6a: Typical Mould Shrinkage Values for PFA

<table>
<thead>
<tr>
<th>Part Thickness</th>
<th>1 mm (0.04”)</th>
<th>2 mm (0.08”)</th>
<th>4 mm (0.16”)</th>
</tr>
</thead>
<tbody>
<tr>
<td>% Shrinkage Parallel to Flow</td>
<td>1.6</td>
<td>2.6</td>
<td>4</td>
</tr>
<tr>
<td>% Shrinkage Perpendicular to Flow</td>
<td>3.8</td>
<td>3.8</td>
<td>4.6</td>
</tr>
</tbody>
</table>

Table 6b: Typical Mould Shrinkage Values for ETFE
Listed below are several issues to consider in addition to the data provided in Tables 6a and 6b.

- **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.

- **Fluorothermoplastics**, 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 mould temperatures.

- In many moulds, some or all of the part dimensions are constrained from shrinking by steps, bosses, and / or cores. Those regions in the mould that are constrained are referred to as fixtured regions. As constrained areas are cooled more in the mould, less shrinkage will be observed in these fixture areas. Increased cooling in the mould can be caused by decreased mould temperatures, decreased part thickness and increased cooling time. Constraining certain regions of the mould 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, or heat treating, is sometimes 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.

Below is an example for annealing of parts, injection moulded from PFA:

- 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 thicknesses.

<table>
<thead>
<tr>
<th>Part Thickness</th>
<th>1.6 mm (0.0625”)</th>
<th>3.2 mm (0.125”)</th>
<th>4.8 mm (0.1875”)</th>
<th>6.4 mm (0.25”)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time at 250 °C (482 °F)</td>
<td>20 min</td>
<td>30 min</td>
<td>45 min</td>
<td>60 min</td>
</tr>
<tr>
<td>% Shrinkage Parallel to Flow</td>
<td>0.4-0.6</td>
<td>0.3-0.5</td>
<td>0.1-0.3</td>
<td>0.1-0.3</td>
</tr>
<tr>
<td>% Shrinkage Perpendicular to Flow</td>
<td>0.6-0.7</td>
<td>0.4-0.5</td>
<td>0.2-0.4</td>
<td>0.1-0.3</td>
</tr>
</tbody>
</table>

Table 7: Annealing Shrinkage of PFA
6. Typical Processing Conditions

6.1 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 or using a lower MFI polymer generally requires higher melt temperatures to reduce the viscosity.

Table 8 shows recommended temperatures for the barrel and the mould. It is recommended to always start at the lower end of the range and find the optimum conditions in going up in temperature to avoid unnecessary degradation of the polymer.

<table>
<thead>
<tr>
<th>Zone</th>
<th>PFA</th>
<th>FEP</th>
<th>ETFE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Centre</td>
<td>350-390 (660-735)</td>
<td>310-350 (590-660)</td>
<td>300 - 320 (570-610)</td>
</tr>
<tr>
<td>Front</td>
<td>360-400 (680-750)</td>
<td>350-400 (660-750)</td>
<td>320 - 350 (610-660)</td>
</tr>
<tr>
<td>Nozzle</td>
<td>360-400 (680-750)</td>
<td>350-400 (660-750)</td>
<td>320 - 350 (610-660)</td>
</tr>
<tr>
<td>Melt</td>
<td>360-400 (680-750)</td>
<td>350-400 (660-750)</td>
<td>320 - 350 (610-660)</td>
</tr>
<tr>
<td>Mould</td>
<td>180-260 (355-500)</td>
<td>180-210 (355-410)</td>
<td>100 - 150 (210-300)</td>
</tr>
</tbody>
</table>

Table 8: Barrel and Mould Temperature Setpoints
Viscosity loss is an indicator that the polymer 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 moulder’s barrel for various times and temperatures. Once these parts are moulded, the melt flow index (MFI) can then be measured.

The contour graph in Figure 12 shows the effect of residence time and temperature on the MFI on the example of 3M™ Dyneon™ Fluoroplastic PFA 6515N. The region shaded in light gray depicts the desirable process window for the material. The regions shaded in dark gray indicate the areas of unfavorable degradation.

Figure 12: Desirable Times and Temperatures
6.2 Mould Temperature

Determination of the mould temperature setpoint should consider part geometry, desired surface finish, stress / warpage, delamination, ejection, shrink and cycle time. Higher mould 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. Recommended mould temperatures are listed in Table 8 (page 31).

6.3 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 Fluorothermoplastics 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 polymer solidifies against the surface of the mould without enough pressure to pack it against the mould wall. Extremely slow injection velocities will leave a rough surface and can result in a short shot.

Typical injection velocities 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.
6.4 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.

6.5 Pack Pressure and Pack Time

Pack/holding 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 375 - 475 bar (5500 - 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.

6.6 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).
In some cases 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. Recommended speed and pressure settings are summarized in Table 9.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>PFA</th>
<th>FEP</th>
<th>ETFE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Screw speed</td>
<td>rpm</td>
<td>50-100</td>
<td>50-100</td>
<td>50-100</td>
</tr>
<tr>
<td>Back pressure</td>
<td>bar (PSI)</td>
<td>50-70 (750-1000)</td>
<td>50-70 (750-1000)</td>
<td>50-70 (750-1000)</td>
</tr>
<tr>
<td>Injection speed</td>
<td>mm/s (in/sec)</td>
<td>3-15 (0.1-0.6)</td>
<td>3-15 (0.1-0.6)</td>
<td>3-15 (0.1-0.6)</td>
</tr>
<tr>
<td>Injection pressure</td>
<td>bar (PSI)</td>
<td>400-800 (5800-11600)</td>
<td>400-800 (5800-11600)</td>
<td>300-600 (4300-8600)</td>
</tr>
<tr>
<td>Pack pressure</td>
<td>bar (PSI)</td>
<td>250-475 (3600-6900)</td>
<td>250-475 (3600-6900)</td>
<td>250-400 (3600-5800)</td>
</tr>
</tbody>
</table>

Table 9: Injection Moulding Setpoints
7. Additional Moulding Considerations

7.1 Delamination

Fluorothermoplastic IM 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 moulding.

Time 1 in Figure 13 shows the velocity profile near the flow front. During fill the material begins to freeze against the mould 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 mould 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 moulded 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 mould temperature and decrease the injection velocity. Lower injection velocities along with higher mould temperatures reduces the chances of forming an unentangled boundary between the frozen layer and the molten material.
Figure 13: Delamination Mechanism

Time 1
- Melt Velocity At Flowfront
- Flow Channel Wall

Time 2
- Frozen Layer Increases
- Molten Material
- Reduced Flow Channel Increases Sheer Rate

Time 3
- Velocity Profile After Sheer Heat Prevents Further Freezing

Additional Moulding Considerations
7.2 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 more information please refer to the “Guide for the Safe Handling of Fluoropolymer Resins”10.

Ventilation

Exhaust hoods should be used to remove any fumes that are generated during the processing of Fluorothermoplastics. It is also desirable to have proper ventilation over all areas where hot parts are placed after being removed from the mould.

Personal protection

During production, the mould, 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.
8. Trouble Shooting

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

<table>
<thead>
<tr>
<th>Problem</th>
<th>Solution</th>
</tr>
</thead>
</table>
| **Short shot** | • Remove restrictions in the material flow path by opening up the nozzle, sprue, runners and gate  
• Increase pack pressure and / or time  
• Increase mould 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** | • Increase decompression distance |
| **Parts sticking in the mould** | • Decrease pack pressure  
• Increase draft on the part  
• Plate the mould with PTFE impregnated nickle plating |
| **Bubbles in the part** (bubbles will expand when a freshly moulded 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 moulded part is heated to near melt temperature) | • Design part with common wall thickness  
• Increase pack pressure  
• Increase mould 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 |

*VPT = velocity to pressure transfer*
9. Appendix

Company References

1. Reiloy Metall GmbH
   Spicher Str. 46-48
   53829 Troisdorf
   Germany
   www.reiloy.com

2. IDM – Inductametals Corp.
   101 West Grand Ave.
   Suite 504
   Chicago, IL 60610
   USA

3. Special Metals Corporation
   3200 Riverside Dr.
   Huntington, WV 25705-1771
   USA
   www.specialmetals.com

4. Wexco Corporation
   1015 Dillard Dr.
   P.O. Box 4297
   Lynchburg, VA 24502
   USA
   www.wexco.com

5. Haynes International
   1020 W. Park Ave.
   P.O. Box 9013
   Kokomo, IN 46904-9013
   USA
   www.haynesintl.com

6. Nordson Xaloy Europe GmbH
   Richard-Wagner-Str. 21
   74172 Neckarsulm
   Germany
   www.nordsonxaloy.com

7. Poly-Plating, Inc.
   2096 Westover Rd.
   Chicopee, MA 01022
   USA
   www.poly-ond.com

8. Bales Mold Service
   2824 Hitchcock Ave.
   Downers Grove, IL 60515
   USA
   www.balesmold.com

9. PSG Plastic Service GmbH
   Pirnaer Str. 12-16
   68309 Mannheim
   Germany
   www.psg-online.de

10. Plastics Europe
    Avenue E. van Nieuwenhuyse 4
     Box 3
     1160 Brussels
     Belgium
     www.plasticseurope.org
Technical Information and Test Data

Technical information and guidance provided by Dyneon personnel is based upon data and testing which is believed to be reliable. Such advice is intended for use by persons with appropriate technical understanding, knowledge and skills relating to PTFE compounds. No license under any Dyneon or third party intellectual rights is granted or implied by virtue of this information.

General recommendations on health and safety in processing, on work hygiene and on measures to be taken in the event of accident are detailed in our material safety data sheets.

You will find further notes on the safe handling of fluoropolymers in the brochure “Guide for the safe handling of Fluoropolymers Resins” by PlasticsEurope, Box 3, B-1160 Brussels, Tel. +32 (2) 676 17 32. The present edition replaces all previous versions. Please make sure and inquire if in doubt whether you have the latest edition.

Important Notice

All information set forth herein is based on our present state of knowledge and is intended to provide general notes regarding products and their uses. It should not therefore be construed as a guarantee of specific properties of the products described or their suitability for a particular application. 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 quality of our products is warranted under our General Terms and Conditions of Sale as now are or hereafter may be in force.