Michael Day Enterprises, Inc.
Engineering Thermoplastics
Solutions. Every Day.

Quick Processing Reference for Nylon 6

This document provides a brief overview highlighting some typical specifications and processing parameters. The following conditions, although generally applicable, are not necessarily optimum for a specific application.

Tooling Specifications

Gating	**Gate thickness should be ¾ of the wall thickness being gated into**

Below shows typical gate thicknesses for typical part thickness and cross sections. These values can be applied when using all types of gates. If the gate is to be enlarged for increased material flow and pressure transfer, focus on increasing thickness rather then width.

Typical Part Thickness	**Gate thickness**
Up to 0.060" (1.5mm)	**0.040" (1.0mm)**
Up to 0.125" (3.2mm)	**0.060" – 0.090" (1.5mm – 2.3mm)**
Up to 0.187" (4.7mm)	**0.090" – 0.125" (2.3mm – 3.2mm)**

Shrinkage	**Please refer to individual data sheets for shrinkage information**

The data sheets provide typical shrinkage values. There are many conditions and factors that can affect part shrinkage, including part design, material processing, location and size of gates. Large variations in cross section thickness can lead to unequal stresses throughout the part and tend to cause differential shrinkage and warpage to become more pronounced.

Prototyping is required to accurately determine precise shrinkage.

Typical Venting Dimensions	**Land of vent – 0.03" – 0.06"** **Width of vent – 0.375" – 0.500"** **Depth of Vent - 0.0005" – 0.001"** **Depth of relief – 0.01" – 0.013**

During the filling of the cavity, the melt has to displace air that is contained in the cavity. If there is nowhere for this air to go, it may compress, and resist the flow of plastic. As the air compresses it will also heat. This may result in a burn line where the plastic and trapped air makes contact.
There are several concepts that may be used for venting including, parting line vents, adding vent pins, ejector pins (flats ground on pins), overflow wells, incorporating inserts at sections that trap air and sintered metal inserts.

Michael Day Enterprises, Inc.
Engineering Thermoplastics
Solutions. Every Day.

Processing Parameters

Screw Design	**Standard general-purpose screw. Minimum L/D of 20:1 with a Compression Ratio of 3:1 being ideal. Significant amount of screw rotation during injection or the inability to hold a cushion could indicate a worn barrel and/or check valve**
Barrel Capacity	**It is recommended that shot size not exceed 75% of barrel capacity. It is not recommended to go below 30% of the barrel capacity as this can lead to extended barrel residence time in the barrel which can, in turn, lead to thermal degradation of the material, part brittleness and discoloration.**
Clamp Pressures	**Cavity Pressure Estimate - 3- 5 tons/in² of projected area** ***Minimum* clamp force required calculation below:** **Part projected (in²) area (measured) X Cavity Pressure Estimate (see above)**
Nozzle	**Nylon Reverse taper nozzle is recommended. This type of nozzle is recommended to minimize material drool and stringing.** **For ease of sprue removal, design the sprue bushing diameter 0.005" – 0.030" (.125mm - .75mm) larger then the nozzle tip orifice diameter.**
Screw RPM	**Screw recovery speeds that will permit screw rotation during 75%-90% of the cooling time are recommended.**
Screw Cushion	**A cushion of 0.100" – 0.250" is recommended. The inability to hold a consistent cushion is usually a sign of a worn non-return valve.**
Temperature Settings	**Unfilled Nylon 6** **Rear – 430 - 480°F (221-249°C)** **Center – 460 - 510 °F (238-266°C)** **Front – 470 – 540°F (243-282°C)** **Nozzle – 465 – 540°F (241-282°C)** **Melt – 470 – 540°F (243– 282°C)** **Reinforced Nylon 6** **Rear – 490 - 540°F (255 - 282°C)** **Center – 500 - 550°F (266-287°C)** **Front – 500– 560°F (266-294°C)** **Nozzle – 510– 560°F (268-294°C)** **Melt – 520-580°F (271 – 306°C)** ***A mold surface temperature in the recommended range may help to improve surface appearance, consistency of mold fill and dimensions and help to achieve the best molded part performance. *Mold Temperature – 160 - 200°F***
Injection speed	**Medium to High – In some cases, slower fill speeds may help in eliminating gate blush.** **A fast injection speed maximizes weld line strength, minimize molded in stresses and also helps to achieve the best surface gloss. Good venting of the cavities is essential to allow a fast fill without burning.**
Back pressure	**25 to 75PSI recommended to insure a consistent shot size and homogeneous melt** ***For glass filled nylons, 50psi is recommended, therefore minimizing the risk of mechanical damage to the glass fibers with consequent loss of part performance.**
Drying	**Nylon materials are hygroscopic and therefore may require drying. The recommended moisture content (by weight) is 0.10 – 0.20%.**

Michael Day Enterprises, Inc.
Engineering Thermoplastics
Solutions. Every Day.

Process Start-Up (Optimizing Fill and Packing)

The following procedure works well with a velocity-controlled injection molding machine. The following conditions, although generally applicable, are not necessarily optimum for a specific application.

1. **Set pack and hold time and pressures to zero**
2. **Set injection pressure to the maximum setting**
3. **Set transfer position to 6 to 7 mm (0.25 to 0.30")**
4. **Set injection time long enough to reach transfer point**
5. **Set injection speed to 0.5 to 1.5 seconds**
6. **Set shot size to achieve a short shot**
7. **Increase shot size until part is 95% filled**
8. **Note injection pressure required to maintain 95% fill of the part**
9. **Set injection pressures to 300 –400 PSI above the injection pressure determined in the previous step; this ensures a velocity controlled filling.**

10) **Set pack and hold pressure to 50% of the injection molding pressure setting**
11) **Set the pack –hold time to 5 – 8 seconds to ensure that the remaining 5% of the part is filled and that there is enough additional material to compensate for shrinkage**
12) **Fine-tune the pack-hold time by running a series of test molding cycles. Weigh each part after each test cycle. Continue to increase the pack-hold time required until the part weight does not increase. This ensures gate freeze off has occurred and that parts have a repeatable, correct weight.**

__Shut down procedure__

The following shut down procedure is recommended:

1. **While molding on cycle, shut the hopper feed slide, empty the hopper.**
2. **Add a quantity of polypropylene and extrude until the barrel is emptied.**
3. **Leave the screw in the "fully" forward position.**
4. **Turn off screw cylinder heaters leaving the nozzle and adaptor heaters on until the barrel temperatures are cool.**
5. **Contamination and the time required for subsequent start-ups may now be reduced.**

__Handling regrind__

The following are recommendations for regrind use:

1. **Typical levels of regrind are 20% - 30%.**
2. **It is recommended to use your regrind back into the process as it is generated.**
3. **Ensure moisture levels are similar to that of the virgin materials.**
4. **Regrind that contains excessive quantities of fines dust-like particles may result in molding issues such as splay or burning.**
5. **If storing regrind, store in moisture proof containers**