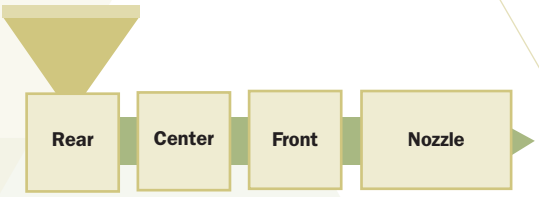
PolyOne

Processing Guide

Global Engineered Materials

Nymax™ 600 Polyamide Compounds

Nymax™ 600 Series Polyamide Compounds are based on engineering resins of nylon 6 chemistry that comprise one of the broadest lines of crystalline compounds. Nymax 600 is offered as modified nylon 6, glass-fiber-reinforced, mineral-reinforced and toughened (impact-modified) nylon 6. With this variety of reinforcements and fillers, Nymax 600 is formulated to meet the strong demands of the automotive, consumer durables, industrial/construction and appliance industries. Nymax 600 is offered in natural and black.



Injection Molding Parameters

Nylon 6	Blends 600 A	Glass-Reinforced GF 600 A	Mineral-Reinforced MF 600 A	Glass-Fiber/Mineral Reinforced GMF 600 A	Impact-Modified 1108/1010
Barrel Temperatures* °C (°F)					
Rear Zone	221 - 249 (430 - 480)	249 - 277 (480 - 530)	260 - 277 (500 - 530)	260 - 277 (500 - 530)	221 - 254 (430 - 490)
Center Zone	238 - 266 (460 - 510)	260 - 288 (500 - 550)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	227 - 260 (440 - 500)
Front Zone	243 - 282 (470 - 540)	271 - 299 (520 - 570)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	238 - 271 (460 - 520)
Nozzle	240 - 280 (465 - 535)	268 - 296 (515 - 565)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	235 - 268 (455 - 515)
Mold Temperature °C (°F)	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)
Pack Pressure	25% - 50% of Injection Pressure	25% - 50% of Injection Pressure	25% - 50% of Injection Pressure	25% - 50% of Injection Pressure	25% - 50% of Injection Pressure
Hold Pressure	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure
Injection Velocity in/s	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0
Back Pressure psi	25 - 100	25 - 100	25 - 100	25 - 100	25 - 100
Screw Speed rpm	30 - 75	30 - 75	30 - 75	30 - 75	30 - 75
Drying Parameters °C (°F)	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)
Cushion in	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250
Screw Compression Ratio	3.0:1	2.5:1	2.5:1	2.5:1	2.5:1
Nozzle Type	Reverse Taper	Reverse Taper	Reverse Taper	Reverse Taper	Reverse Taper
Clamp Pressure	2 - 3 Tons/in^2	3 - 5 Tons/in^2	3 - 5 Tons/in^2	3 - 5 Tons/in^2	3 - 5 Tons/in^2
Cooling Time Seconds	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40
Screw Type	Bimetallic Screw L/D Ratio = 20:1	Bimetallic Screw L/D Ratio = 20:1	Bimetallic Screw L/D Ratio = 20:1	Bimetallic Screw L/D Ratio = 20:1	Bimetallic Screw L/D Ratio = 20:1

STARTUP & SHUTDOWN	RECOMMENDATIONS
Drying	Drying Nymax 600 Series is recommended at 180°F (82°C) for 4 hours. The recommended moisture level by weight for unreinforced and less than 20% reinforced compounds is 0.10%-0.20%. For compounds that are greater than 20% reinforced, moisture level of 0.06%-0.12% is recommended. Moisture levels below 0.02% are not recommended.
Purge Compound	Polypropylene
Coloring	Contact your PolyOne representative.
Recycling	Nymax 600 Series is fully recyclable. Conventional granulators with sharp blades should be used. Consistent regrind usage of up to 20% is permissible. Excessive fines or dust-like particles should be avoided. Drying regrind for 4 hours at 180°F (82°C) is recommended.

MOLD DESIGN	RECOMMENDATIONS
Tool Steel	1. For unreinforced nylon 6 compounds, P20 tool steel is recommended. 2. For reinforced nylon 6 compounds, S7/H13/420 tool steel is recommended to help reduce wear.
Gates	1. All types of gates can be used such as pin, fan, tunnel, tab and edge gates. Gate type should be selected based on location and part geometry. 2. Gate diameters should be equivalent to 50%-80% of the average wall thickness of the part to be injected. 3. A land length of 0.040" (1.0mm) is recommended.
Runners	1. Full-round or modified trapezoid runners are the best design and provide the least surface to cross-section ratio. Half-round or standard trapezoid runners are not recommended. 2. Only naturally balanced runner systems ("H" pattern) are recommended. 3. Each 90° bend in the system should step down in size. 4. Vents should be placed at the intersection of each 90° bend off of the cold slug well and vented to atmosphere. 5. Hot runner molds are acceptable and should be sized by the manufacturer. Externally heated manifolds are recommended.
Cold Slug Wells	1. Place cold slug wells at the base of the sprue to capture the cold material first emerging from the nozzle. 2. Place cold slug wells at every 90° bend in the runner system. 3. Well depths approximately 2-3 times the diameter of the runner provide best results.
Venting	1. Place vents at the end of fill and anywhere potential knit/weld lines will occur. 2. All vents need to be vented to atmosphere. 3. For unreinforced nylon 6 compounds, cut vent depths to 0.0005"-0.001" with a minimum 0.030" land length. Increase the vent depth to 0.010" at 0.100" away from the cavity and vent to atmosphere. 4. For reinforced nylon 6 compounds (greater than 20% filler), cut vent depths to 0.001"-0.002" with a minimum 0.030" land length. Increase the vent depth to 0.010" at 0.100" away from the cavity and vent to atmosphere. 5. Vents should be placed at the intersection of each 90° bend in the runner system off of the cold slug well and vented to atmosphere.
Draft Angle	1. Draft angle should be 1/2°-1° per side. Additional draft may be required for grained/textured surfaces.

TROUBLESHOOTING RECOMMENDATIONS

Problem	Cause	Solution
Incomplete Fill	Melt and/or mold too cold	1. Increase nozzle and barrel temperatures 2. Increase mold temperature 3. Increase injection rate 4. Increase pack and hold pressure 5. Increase nozzle tip diameter 6. Check thermocouples and heater bands
	Mold design	1. Enlarge or widen vents and increase number of vents 2. Check that vents are unplugged 3. Check that gates are unplugged 4. Enlarge gates and/or runners 5. Perform short shots to determine fill pattern and verify proper vent location 6. Increase wall thickness to move gas trap to parting line
	Shot size	1. Increase shot size 2. Increase cushion
Brittleness	Low melt temperature	1. Increase melt temperature 2. Increase injection rate 3. Measure melt temperature with pyrometer
	Degraded/Overheated material	1. Decrease melt temperature 2. Decrease back pressure 3. Use smaller barrel/excessive residence time
	Gate location and/or size	1. Relocate gate to nonstress area 2. Increase gate size to allow higher flow rate and lower molded-in stress
Fibers/Minerals on Surface or Uneven Surface Appearance	Melt temperature too low	1. Increase melt temperature 2. Increase mold temperature 3. Increase injection speed
	Insufficient packing	1. Increase pack and hold pressure, and time 2. Increase shot size
Sink Marks	Part geometry too thick	1. Reduce wall thickness 2. Reduce rib thickness
	Melt too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperature
	Insufficient material volume	1. Increase shot size 2. Increase injection rate 3. Increase packing pressure 4. Increase gate size
Flash	Injection pressure too high	1. Decrease injection pressure 2. Increase clamp pressure 3. Decrease injection rate 4. Increase transfer position
	Excess material volume	1. Decrease pack pressure 2. Decrease shot size 3. Decrease injection rate
	Melt and/or mold too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperature 3. Decrease screw speed

TROUBLESHOOTING RECOMMENDATIONS

Problem	Cause	Solution
Excessive Shrink	Too much orientation	1. Increase packing time and pressure 2. Increase hold pressure 3. Decrease melt temperature 4. Decrease mold temperature 5. Decrease injection speed 6. Decrease screw rpm 7. Increase venting 8. Increase cooling time
Not Enough Shrink	Too little orientation	1. Decrease packing pressure and time 2. Decrease hold pressure 3. Increase melt temperature 4. Increase mold temperature 5. Increase injection speed 6. Increase screw rpm 7. Decrease cooling time
Burning	Melt and/or mold too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperature 3. Decrease injection rate
	Mold design	1. Clean, widen and increase number of vents 2. Increase gate size or number of gates.
	Moisture	1. Verify material is dried at proper conditions
Nozzle Drool	Nozzle temperature too hot	1. Decrease nozzle temperature 2. Decrease back pressure 3. Increase screw decompression 4. Verify material has been dried at proper conditions
Weld Lines	Melt front temperatures are too low	1. Increase pack and hold pressure 2. Increase melt temperature 3. Increase vent width and locations 4. Increase injection rate 5. Increase mold temperature
	Mold design	1. Decrease injection rate 2. Increase gate size 3. Perform short shots to determine fill pattern and verify proper vent location 4. Add vents and/or false ejector pin 5. Move gate location
Warp	Excessive orientation	1. Increase cooling time 2. Increase melt temperature 3. Decrease injection pressure and injection rate
	Mold design	1. Increase number of gates
Sticking in Mold	Cavities are overpacked	1. Decrease injection rate and pressure 2. Decrease pack and hold pressure 3. Decrease nozzle and barrel temperatures 4. Decrease mold temperature 5. Increase cooling time
	Mold design	1. Increase draft angle
	Part is too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperature 3. Increase cooling time

For questions or issues, please call Global Engineered Materials Technical Support at: 440.930.1000