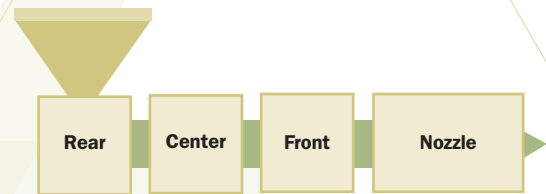




# Nymax™ 600 Polyamide Compounds

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
<b>Barrel Temperatures* °C (°F)</b>					
<b>Rear Zone</b>	221 - 249 (430 - 480)	249 - 277 (480 - 530)	260 - 277 (500 - 530)	260 - 277 (500 - 530)	221 - 254 (430 - 490)
<b>Center Zone</b>	238 - 266 (460 - 510)	260 - 288 (500 - 550)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	227 - 260 (440 - 500)
<b>Front Zone</b>	243 - 282 (470 - 540)	271 - 299 (520 - 570)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	238 - 271 (460 - 520)
<b>Nozzle</b>	240 - 280 (465 - 535)	268 - 296 (515 - 565)	274 - 288 (525 - 550)	274 - 288 (525 - 550)	235 - 268 (455 - 515)
<b>Mold Temperature °C (°F)</b>	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)	49 - 93 (120 - 200)
<b>Pack Pressure</b>	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
<b>Hold Pressure</b>	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure	30% of Injection Pressure
<b>Injection Velocity in/s</b>	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0	2.0 - 3.0
<b>Back Pressure psi</b>	25 - 100	25 - 100	25 - 100	25 - 100	25 - 100
<b>Screw Speed rpm</b>	30 - 75	30 - 75	30 - 75	30 - 75	30 - 75
<b>Drying Parameters °C (°F)</b>	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)	4hrs @ 82 (180)
<b>Cushion in</b>	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250
<b>Screw Compression Ratio</b>	3.0:1	2.5:1	2.5:1	2.5:1	2.5:1
<b>Nozzle Type</b>	Reverse Taper	Reverse Taper	Reverse Taper	Reverse Taper	Reverse Taper
<b>Clamp Pressure</b>	2 - 3 Tons/in <sup>2</sup>	3 - 5 Tons/in <sup>2</sup>	3 - 5 Tons/in <sup>2</sup>	3 - 5 Tons/in <sup>2</sup>	3 - 5 Tons/in <sup>2</sup>
<b>Cooling Time Seconds</b>	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40
<b>Screw Type</b>	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
<b>Drying</b>	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.
<b>Purge Compound</b>	Polypropylene
<b>Coloring</b>	Contact your PolyOne representative.
<b>Recycling</b>	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
<b>Tool Steel</b>	<ol style="list-style-type: none"> <li>1. For unreinforced nylon 6 compounds, P20 tool steel is recommended.</li> <li>2. For reinforced nylon 6 compounds, S7/H13/420 tool steel is recommended to help reduce wear.</li> </ol>
<b>Gates</b>	<ol style="list-style-type: none"> <li>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.</li> <li>2. Gate diameters should be equivalent to 50%-80% of the average wall thickness of the part to be injected.</li> <li>3. A land length of 0.040" (1.0mm) is recommended.</li> </ol>
<b>Runners</b>	<ol style="list-style-type: none"> <li>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.</li> <li>2. Only naturally balanced runner systems ("H" pattern) are recommended.</li> <li>3. Each 90° bend in the system should step down in size.</li> <li>4. Vents should be placed at the intersection of each 90° bend off of the cold slug well and vented to atmosphere.</li> <li>5. Hot runner molds are acceptable and should be sized by the manufacturer. Externally heated manifolds are recommended.</li> </ol>
<b>Cold Slug Wells</b>	<ol style="list-style-type: none"> <li>1. Place cold slug wells at the base of the sprue to capture the cold material first emerging from the nozzle.</li> <li>2. Place cold slug wells at every 90° bend in the runner system.</li> <li>3. Well depths approximately 2-3 times the diameter of the runner provide best results.</li> </ol>
<b>Venting</b>	<ol style="list-style-type: none"> <li>1. Place vents at the end of fill and anywhere potential knit/weld lines will occur.</li> <li>2. All vents need to be vented to atmosphere.</li> <li>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.</li> <li>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.</li> <li>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.</li> </ol>
<b>Draft Angle</b>	<ol style="list-style-type: none"> <li>1. Draft angle should be 1/2°-1° per side. Additional draft may be required for grained/textured surfaces.</li> </ol>

## TROUBLESHOOTING RECOMMENDATIONS

Problem	Cause	Solution
<b>Incomplete Fill</b>	Melt and/or mold too cold	<ol style="list-style-type: none"> <li>1. Increase nozzle and barrel temperatures</li> <li>2. Increase mold temperature</li> <li>3. Increase injection rate</li> <li>4. Increase pack and hold pressure</li> <li>5. Increase nozzle tip diameter</li> <li>6. Check thermocouples and heater bands</li> </ol>
	Mold design	<ol style="list-style-type: none"> <li>1. Enlarge or widen vents and increase number of vents</li> <li>2. Check that vents are unplugged</li> <li>3. Check that gates are unplugged</li> <li>4. Enlarge gates and/or runners</li> <li>5. Perform short shots to determine fill pattern and verify proper vent location</li> <li>6. Increase wall thickness to move gas trap to parting line</li> </ol>
	Shot size	<ol style="list-style-type: none"> <li>1. Increase shot size</li> <li>2. Increase cushion</li> </ol>
<b>Brittleness</b>	Low melt temperature	<ol style="list-style-type: none"> <li>1. Increase melt temperature</li> <li>2. Increase injection rate</li> <li>3. Measure melt temperature with pyrometer</li> </ol>
	Degraded/Overheated material	<ol style="list-style-type: none"> <li>1. Decrease melt temperature</li> <li>2. Decrease back pressure</li> <li>3. Use smaller barrel/excessive residence time</li> </ol>
	Gate location and/or size	<ol style="list-style-type: none"> <li>1. Relocate gate to nonstress area</li> <li>2. Increase gate size to allow higher flow rate and lower molded-in stress</li> </ol>
<b>Fibers/Minerals on Surface or Uneven Surface Appearance</b>	Melt temperature too low	<ol style="list-style-type: none"> <li>1. Increase melt temperature</li> <li>2. Increase mold temperature</li> <li>3. Increase injection speed</li> </ol>
	Insufficient packing	<ol style="list-style-type: none"> <li>1. Increase pack and hold pressure, and time</li> <li>2. Increase shot size</li> </ol>
<b>Sink Marks</b>	Part geometry too thick	<ol style="list-style-type: none"> <li>1. Reduce wall thickness</li> <li>2. Reduce rib thickness</li> </ol>
	Melt too hot	<ol style="list-style-type: none"> <li>1. Decrease nozzle and barrel temperatures</li> <li>2. Decrease mold temperature</li> </ol>
	Insufficient material volume	<ol style="list-style-type: none"> <li>1. Increase shot size</li> <li>2. Increase injection rate</li> <li>3. Increase packing pressure</li> <li>4. Increase gate size</li> </ol>
<b>Flash</b>	Injection pressure too high	<ol style="list-style-type: none"> <li>1. Decrease injection pressure</li> <li>2. Increase clamp pressure</li> <li>3. Decrease injection rate</li> <li>4. Increase transfer position</li> </ol>
	Excess material volume	<ol style="list-style-type: none"> <li>1. Decrease pack pressure</li> <li>2. Decrease shot size</li> <li>3. Decrease injection rate</li> </ol>
	Melt and/or mold too hot	<ol style="list-style-type: none"> <li>1. Decrease nozzle and barrel temperatures</li> <li>2. Decrease mold temperature</li> <li>3. Decrease screw speed</li> </ol>

## TROUBLESHOOTING RECOMMENDATIONS

Problem	Cause	Solution
<b>Excessive Shrink</b>	Too much orientation	<ol style="list-style-type: none"> <li>1. Increase packing time and pressure</li> <li>2. Increase hold pressure</li> <li>3. Decrease melt temperature</li> <li>4. Decrease mold temperature</li> <li>5. Decrease injection speed</li> <li>6. Decrease screw rpm</li> <li>7. Increase venting</li> <li>8. Increase cooling time</li> </ol>
<b>Not Enough Shrink</b>	Too little orientation	<ol style="list-style-type: none"> <li>1. Decrease packing pressure and time</li> <li>2. Decrease hold pressure</li> <li>3. Increase melt temperature</li> <li>4. Increase mold temperature</li> <li>5. Increase injection speed</li> <li>6. Increase screw rpm</li> <li>7. Decrease cooling time</li> </ol>
<b>Burning</b>	Melt and/or mold too hot	<ol style="list-style-type: none"> <li>1. Decrease nozzle and barrel temperatures</li> <li>2. Decrease mold temperature</li> <li>3. Decrease injection rate</li> </ol>
	Mold design	<ol style="list-style-type: none"> <li>1. Clean, widen and increase number of vents</li> <li>2. Increase gate size or number of gates.</li> </ol>
	Moisture	<ol style="list-style-type: none"> <li>1. Verify material is dried at proper conditions</li> </ol>
<b>Nozzle Drool</b>	Nozzle temperature too hot	<ol style="list-style-type: none"> <li>1. Decrease nozzle temperature</li> <li>2. Decrease back pressure</li> <li>3. Increase screw decompression</li> <li>4. Verify material has been dried at proper conditions</li> </ol>
<b>Weld Lines</b>	Melt front temperatures are too low	<ol style="list-style-type: none"> <li>1. Increase pack and hold pressure</li> <li>2. Increase melt temperature</li> <li>3. Increase vent width and locations</li> <li>4. Increase injection rate</li> <li>5. Increase mold temperature</li> </ol>
	Mold design	<ol style="list-style-type: none"> <li>1. Decrease injection rate</li> <li>2. Increase gate size</li> <li>3. Perform short shots to determine fill pattern and verify proper vent location</li> <li>4. Add vents and/or false ejector pin</li> <li>5. Move gate location</li> </ol>
<b>Warp</b>	Excessive orientation	<ol style="list-style-type: none"> <li>1. Increase cooling time</li> <li>2. Increase melt temperature</li> <li>3. Decrease injection pressure and injection rate</li> </ol>
	Mold design	<ol style="list-style-type: none"> <li>1. Increase number of gates</li> </ol>
<b>Sticking in Mold</b>	Cavities are overpacked	<ol style="list-style-type: none"> <li>1. Decrease injection rate and pressure</li> <li>2. Decrease pack and hold pressure</li> <li>3. Decrease nozzle and barrel temperatures</li> <li>4. Decrease mold temperature</li> <li>5. Increase cooling time</li> </ol>
	Mold design	<ol style="list-style-type: none"> <li>1. Increase draft angle</li> </ol>
	Part is too hot	<ol style="list-style-type: none"> <li>1. Decrease nozzle and barrel temperatures</li> <li>2. Decrease mold temperature</li> <li>3. Increase cooling time</li> </ol>

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