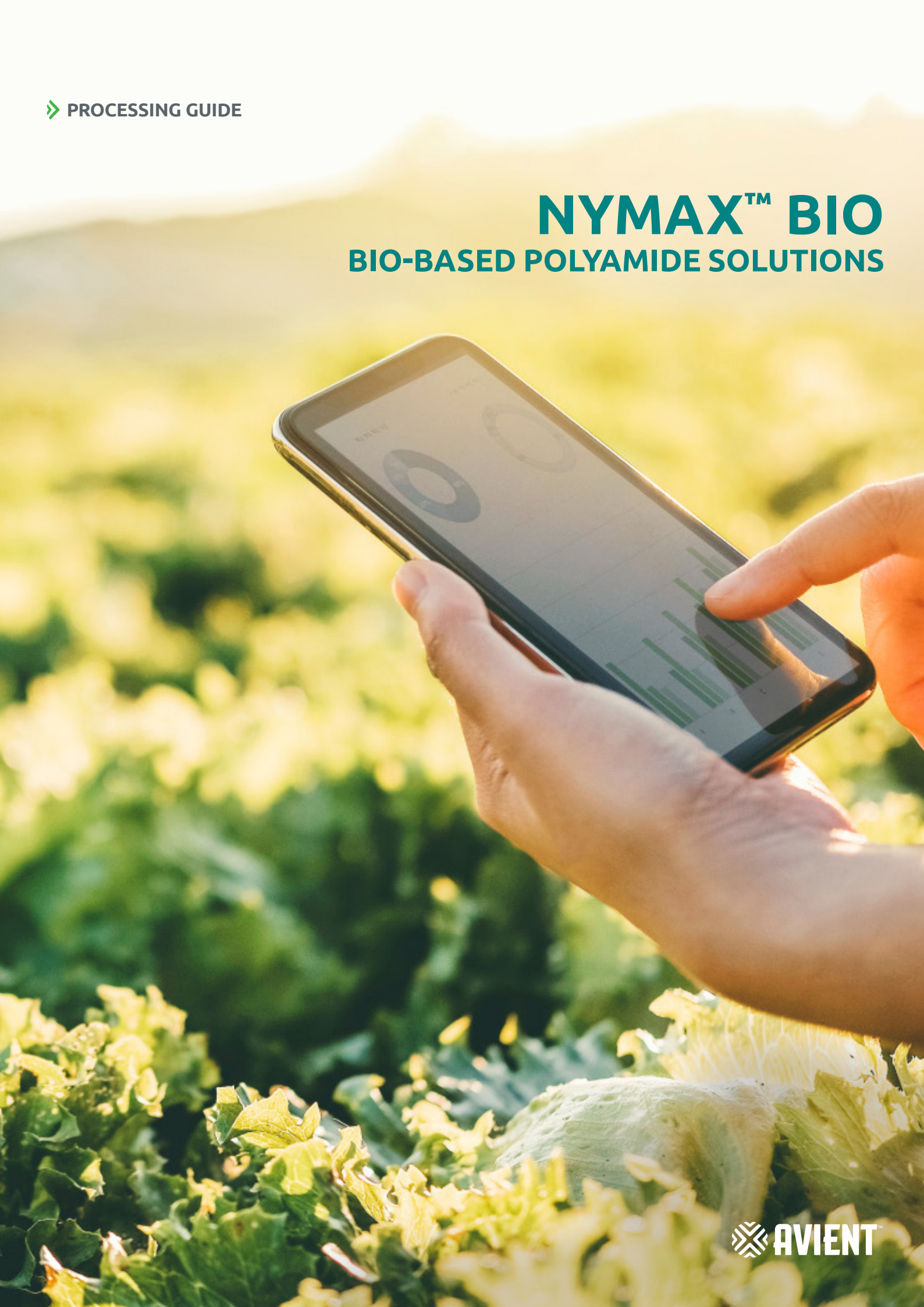
PROCESSING GUIDE

NYMAX™ BIO

BIO-BASED POLYAMIDE SOLUTIONS



AVIENT

Formulated in glass fiber-filled or unfilled materials, **Nymax™ BIO** formulations utilize between 16 and 47 percent natural filler from renewable plant-based raw materials, reducing the carbon footprint value significantly at the beginning of the product lifecycle. These highly engineered formulations deliver comparable performance to traditional PA66 glass fiber-filled materials, plus the bio-derived solutions offer lower warpage and have excellent surface appearance and colorability. Nymax BIO low water absorption PA formulations provide excellent dimensional stability and property retention after conditioning, solving the problem of water uptake (hygroscopy) for finished parts.

Parameters	NM5600-8002 RS	NM5600-8004 RS
Drying Temperature	100–120°C	100–130°C
Drying Time	4–6 hours	4–6 hours
Barrel Temperature	°C	°C
Rear Zone	250–270	260–270
Central Zone	260–280	270–290
Front Zone	270–295	290–305
Nozzle	290–300	290–300
Mold Temperature	80–100	80–100
Screw Speed	Moderate	Moderate
Back Pressure	3–10bar	3–10bar
Cushion stroke	5–10% of plasticizing stroke	5–10% plasticizing of stroke
Injection Speed	Medium	Medium
Injection Pressure	Medium–high	Medium–high
Holding Pressure	30–50% of injection pressure	20–40% of injection pressure
Screw Type	General purpose	General purpose
Screw L/D	20:1	20:1
Screw Compression Ratio	2.5:1	2.5:1
Non-return Check Valve	Free flow check ring	Free flow check ring
Nozzle Type	Reverse taper	Reverse taper
Barrel Capacity	30–80% of barrel should be used	

Start Up & Shut Down	Recommendations
Purge Compound	1. 2–3 melt flow PP or purging compound. PC or PA also recommended
Regrind	1. Regrind is not suggested—glass fiber reinforce material will lose strength after regrind
Mold Design	**Recommendations**
Gates	1. All types of gates can be used such as pin, fan, tunnel, tab and edge gates. Gate type should be select 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 inject. 3. A land length of 0.040” (1.0mm) is recommend
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 the 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 the best results.
Vents	1. Place vents at the end of fill and anywhere potential knit/weld lines will occur. 2. All vents need to be vented to the atmosphere. 3. 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	1. Draft angle should be 1/2° to 1° per side. Additional draft may be required for grained/textured surfaces.

TROUBLESHOOTING

Implement recommended solutions one at a time in the order specified until problem is resolved.

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. 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. Adjust transfer positon to 98% full 3. Increase cushion
Brittleness	Degraded/ overheated material	1. Decrease melt temperature 2. Decrease back pressure 3. Use smaller barrel 4. Decrease injection speed
	Gate location and/or size	1. Relocate gate to non-stress area 2. Increase gate size to allow higher flow rate and lower molded in stress
	Wet material	1. Check moisture of material to ensure it is within the recommended moisture percentage for molding
Splay	Melt temperature too low	1. Increase melt temperature 2. Increase mold temperature 3. Increase injection speed
	Wet material	1. Check moisture of material to ensure it is within the recommended moisture percentage for molding
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
Shrink	Too much shrink	1. Increase cooling time 2. Decrease mold temperature
	Too little shrink	1. Decrease cooling time 2. Increase mold temperature

Problem	Cause	Solution
Flash	Injection pressure too high	1. Decrease injection pressure 2. Increase clamp pressure 3. Decrease injection speed 4. Increase transfer position
	Excess material volume	1. Adjust transfer position 2. Decrease pack pressure 3. Decrease shot size 4. Decrease injection speed
	Melt and/or mold too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperatures
	Loose clamp	1. Reset mold height 2. Increase clamp tonnage
Burning	Melt and/or mold too hot	1. Decrease nozzle and barrel temperatures 2. Decrease mold temperature 3. Decrease injection speed 4. Reduce decompression
	Mold design	1. Clean, widen and increase number of vents 2. Increase gate size to reduce shear
	Wet material	1. Verify material is dried at proper condition
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
	Incorrect nozzle	1. Use reverse taper nozzle
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 speed 5. Increase mold temperature
	Mold design	1. Increase gate size 2. Identify end of fill pattern and verify proper vent location 3. Add vents or increase vent width 4. Move gate location
Warp	Melt front temperatures are too low	1. Increase melt temp 2. Reduce injection speed 3. Increase pack pressure 4. Increase pack time 5. Decrease mold temperature 6. Increase cool time
	Mold design	1. Non-uniform mold cooling
	Part design	1. Non-uniform wall thickness
	Thermolator incorrect temperature	1. Check settings 2. Inspect thermocouple

Problem	Cause	Solution
Sticking in mold	Cavities are over packed	1. Decrease injection speed and pressure 2. Decrease hold pressure 3. Adjust transfer position 4. Decrease nozzle and barrel temperatures 5. Decrease mold temperature 6. Increase cooling time
	Mold design	1. Increase draft angle 2. Polish cores in direction of ejection
	Part is too hot	1. Decrease barrel temperatures 2. Decrease mold temperature 3. Increase cooling time
Black specks	Contamination	1. Purge machine
	Degradation	1. Reduce melt temperature 2. Reduce screw speed 3. Reduce back pressure
	Machine related	1. Check for wear on screw, barrel or check ring
Delamination	Process related	1. Increase melt temperature 2. Decrease injection speed 3. Purge barrel to eliminate material contamination
	Mold design	1. Reduce sharp corners in material flow path 2. Increase venting
Discoloration	Oversheared material	1. Decrease melt temperature 2. Decrease injection speed 3. Reduce residence time
	Mold design	1. Increase gate sizing
	Dry material	1. Check moisture of material to ensure it is within the recommended moisture percentage for molding

1.844.4AVIENT
www.avient.com