PROCESSING GUIDE

ARTISAN™ PRE-COLORED THERMOPLASTICS

NYLON (PA6) FORMULATIONS

AVIENT

Artisan™ Pre-Colored Thermoplastics

Artisan™ thermoplastics are vibrant, pre-colored high-gloss, scratch resistant engineered polymers formulated to replace paint for superior results. The impact-modified PA6 grade offers minimized scratch appearance compared to painted plastic and improves corrosion resistance when replacing painted metal parts. By removing the painting process, these formulations also offer additional sustainable benefits: energy use is reduced and VOCs are eliminated. Outdoor consumer applications where superior aesthetics, chemical resistance, UV stability, and high performance are critical, such as powersports, lawn and garden, and marine body panels and housings, are suitable candidates for customized Artisan formulations.

Processing Guidelines

Base Resin	PA6	
Barrel Temperatures	°F	°C
Rear Zone	440–470	227–243
Center Zone	450–480	232–249
Front Zone	460–490	238–254
Nozzle	470–500	243–260
Melt Temperature	470–500	243–260
Mold Temperature	120–180	49–82
Pack and Hold Pressure	50–75% of Injection Pressure	
Injection Velocity	0.5 in/s–3 in/s	13 mm/s–76 mm/s
Back Pressure	50–100 psi	3.4–6.9 Bar
Screw Speed	30–70 rpm	
Cushion	0.25 in	6.35 mm
Drying Parameters	2–4 Hours @ 180°F	2–4 Hours @ 82°C
Moisture % Allowable	0.08–0.18%*	
Screw Type	General Purpose Screw	
Screw Compression Ratio	2.0:1– 2.5:1	
Screw L/D	20:1	
Non-return Check Valve	Free Flow Check Ring	
Nozzle Type	Reverse Taper	
Clamp Pressure	2–3 Tons/in^2	
Barrel Capacity	30-80% of barrel should be used	

* Avient suggests measuring moisture using a Karl Fischer method or Vapor Pro® moisture analyzer that titrates only for moisture.
Vapor Pro® is a registered trademark of Arizona Instrument LLC.

Start Up & Shut Down	Recommendations
Purge Compound	2–3 melt flow PP or purging compound. HDPE is not recommended for purging as it can cause delamination or lead to black specks.
Regrind	Regrind is not suggested. Can cause issues with color variation, surface defects, loss of properties and may affect the weatherability.
Mold Design	**Recommendations**
Gates	1. Many different types of gates can be used such as pin, fan, tunnel, tab and edge gates 2. Gate thickness should be 50–75% of wall thickness 3. Avoid gating into thin part region
Runners	1. Full-round runners or modified trapezoid runners are the best designs 2. Half-round runners are not recommended 3. Only naturally balanced runner systems (“H” pattern) are recommended 4. Runner diameters should not be less than the part thickness 5. Runner diameter should be 1.5x the part thickness 6. Step each 90° bend in the system down in size 7. Place vents at each 90° intersection and vent to atmosphere 8. Hot runner molds are acceptable and should be sized by the manufacturer
Cold Slug Wells	1. Place these wells at the base of the sprue to capture the cold material first emerging from the nozzle 2. Place wells at every 90° bend in the runner system 3. Well depths approximately 2.5 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 atmosphere 3. For circular parts, full perimeter venting is recommended 4. Cut vent depths to 0.0007″–0.0015″
Draft Angle	Maintain a minimum draft angle of 1° per side

Troubleshooting Recommendations

Problem	Cause	Solution
Incomplete Fill	Melt and/or mold too cold	• Increase nozzle and barrel temperatures • Increase mold temperature • Increase injection rate • Check thermocouples and heater bands
	Shot size	• Increase shot size • Adjust transfer position to 98% full • Increase cushion
	Mold design	• Enlarge or widen vents and increase number of vents • Check that vents are unplugged • Check that gates are unplugged • Enlarge gates and/or runners • Perform short shots to determine fill pattern and verify proper vent location • Increase wall thickness to move gas trap to parting line
Brittleness	Degraded/overheated material	• Decrease melt temperature • Decrease back pressure • Use smaller barrel • Decrease injection speed
	Gate location and/or size	• Relocate gate to non-stress area • Increase gate size to allow higher flow rate and lower molded-in stress
	Wet material	• Check moisture. If material is not in the recommended moisture percentage for molding, dry material until it is in the acceptable range for molding.
Splay	Melt temperature too low	• Increase melt temperature • Increase mold temperature • Increase injection speed
	Wet material	• Check moisture. If material is not in the recommended moisture percentage for molding, dry material until it is in the acceptable range for molding.
Sink Marks	Melt too hot	• Decrease nozzle and barrel temperatures • Decrease mold temperature
	Insufficient material volume	• Adjust transfer position • Increase shot size • Increase injection rate • Increase packing pressure
	Part geometry too thick	• Reduce wall thickness • Reduce rib thickness

Troubleshooting Recommendations

Problem	Cause	Solution
Flash	Injection pressure too high	• Decrease injection pressure • Increase clamp pressure • Decrease injection rate • Increase transfer position
	Excess material volume	• Adjust transfer position • Decrease pack pressure • Decrease shot size • Decrease injection rate
	Melt or mold too hot	• Decrease nozzle and barrel temperatures • Decrease mold temperature
	Loose clamp	• Reset mold height • Increase clamp tonnage
Shrink	Too much shrink	• Increase cooling time • Decrease mold temperature
	Too little shrink	• Decrease cooling time • Increase mold temperature
Burning	Process related	• Decrease nozzle and barrel temperatures • Decrease mold temperature • Decrease injection rate • Reduce decompression
	Mold design	• Clean, widen and increase number of vents • Increase gate size to reduce shear
	Wet material	• Verify material is dried at proper condition
Nozzle Drool	Nozzle temperature too hot	• Decrease nozzle temperature • Decrease back pressure • Increase screw decompression • Verify material has been dried at proper conditions
	Incorrect nozzle	• Use reverse taper tip
Weld Lines	Melt front temperatures are too low	• Increase injection rate • Increase pack and hold pressure • Increase melt temperature • Increase mold temperature
	Mold design	• Increase gate size • Identify end of fill pattern and verify proper vent location • Add vents or increase vent width • Move gate location

Troubleshooting Recommendations

Problem	Cause	Solution
Warp	Process related	• Increase melt temp • Reduce injection speed • Increase pack pressure • Increase pack time • Decrease mold temperature • Increase cool time
	Mold design	• Non-uniform mold cooling
	Part design	• Non-uniform wall thickness
	Thermolator incorrect temperature	• Check settings • Inspect thermocouple
Sticking in Mold	Overfilled cavity	• Decrease injection rate and pressure • Decrease hold pressure • Adjust transfer position • Decrease nozzle and barrel temperatures • Decrease mold temperature • Decrease cooling time
	Part too hot	• Decrease barrel temperature • Decrease mold temperature • Increase cooling time
	Mold design	• Increase draft angle • Polish cores in direction of ejection
Black Specks	Contamination	• Purge machine
	Degradation	• Reduce melt temperature • Reduce screw speed • Reduce back pressure
	Machine related	• Check for wear on screw, barrel or check ring
Delamination	Process related	• Increase melt temperature • Decrease injection speed • Purge barrel to eliminate material contamination
	Mold design	• Reduce sharp corners in material flow path • Increase venting

Troubleshooting Recommendations

Problem	Cause	Solution
Discoloration	Oversheared material	• Decrease melt temperature • Decrease injection speed • Reduce residence time
	Mold design	• Increase gate sizing
	Dry material	• Check moisture of material to ensure it is within the recommended moisture percentage for molding

1.844.4AVIENT
www.avient.com

AVIENT™