PROCESSING GUIDE

Gravi-Tech™
Density Modified Formulations

AVIENT™

GRAVI-TECH™

Density Modified Formulations

Gravi-Tech™ polymer-metal composites are high-density materials developed as thermoplastic-based alternatives to lead and other traditional metals. These materials have been formulated using select metallic fillers and engineered thermoplastic resins to have densities similar to actual metals, while also providing the design flexibility and processing ease of conventional thermoplastics.

Injection Molding Parameters

These recommendations are regarded as a general starting point. Every molding machine is different in actual performance. Small quantities should be tested before large quantities are used. Actual temperatures should be measured using a pyrometer.

Base Resin	ABS	PA	PBT	
Barrel Temperatures °F (°C)				
Rear Zone	400–475 (200–250)	480–540 (248–282)	480–520 (250–270)	
Center Zone	410–480 (205–253)	490–550 (254–287)	485–525 (251–273)	
Front Zone	420–490 (210–257)	500–560 (260–293)	490–530 (254–276)	
Nozzle	425–500 (215–260)	510–570 (265–298)	490–540 (254–282)	
Melt Temperature	425–515 (215–270)	510–570 (265–298)	490–540 (254–282)	
Mold Temperature	140–200 (60–90)	150–300 (65–148)	140–280 (60–137)	
Drying Parameters	190 (90) 2–4 Hours 0.01%–0.15%	180 (82) 4–5 Hours 0.10%–0.20%	275 (135) 3–4 Hours 0.02%–0.04%	
Nozzle Type	General Purpose	Nylon or Reverse Taper	General Purpose	
Injection Velocity[1]	2–5 in/sec; 50–127 mm/sec			
Injection Pressure	Medium to high (may range from 1,000–4,000 psi; 7-28 Mpa)			
Back Pressure	50–100 psi; 0.3–0.7 Mpa			
Screw Speed	25–75 RPM			
Cushion	0.125–0.250 in; 3–6 mm			
Screw Compression Ratio[2]	2.0:1–2.5:1			

Comments

1. A higher injection velocity is needed when processing Gravi-Tech due to the thermal conductivity of the material.

2. General purpose screws work well.

	PC	PE	PEEK	PP	PPS
	480–570 (250–300)	400–445 (200–230)	660–700 (350–475)	400–440 (200–225)	520–600 (270–300)
	500–580 (260–305)	410–455 (207–237)	670–710 (357–385)	410–450 (205–230)	550–610 (285–310)
	515–590 (267–310)	420–465 (213–243)	680–720 (363–400)	420–455 (215–235)	570–620 (300–320)
	530–600 (275–315)	430–475 (220–250)	700–730 (370–395)	430–460 (220–240)	610–620 (320–325)
	530–615 (275–325)	430–495 (220–260)	700–725 (370–385)	430–475 (220–250)	610–635 (320–335)
	160–240 (70–115)	80–140 (25–60)	300–400 (150–200)	80–140 (25–60)	190–300 (90–150)
	250 (125) 3–4 Hours 0.02%	160 (70) 2 Hours	300 (150) 3–4 Hours 0.10%	160 (70) 2 Hours	280 (135) 2–3 Hours 0.01%–0.20%
	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose

These recommendations are regarded as a general starting point. The use of regrind can affect some parameters. Wear generally occurs around the gate; thus inserts are recommended for large-volume production. Elongated dwell time in the barrel may cause degradation; 20% to 40% of barrel capacity per shot is optimum. Low MFI Polypropylene flush is recommended between production runs.

Mold Design	Recommendations
Gates	1. Many different types of gates can be used such as edge, tab, fan, and tunnel gates. Gate type should be selected based on location and part geometry. 2. Gate diameters should be no less than 2.54 mm (0.100"). 3. Land lengths of 0.50mm–0.90mm (0.020"–0.035") are typically recommended.
Runners	1. Full-round runners or a modified trapezoid runner are the best designs. Half-round runners are not recommended. 2. Only naturally balanced runner systems (“H” pattern) are recommended. 3. Runner diameters should be no less than 3.175mm (0.125”). 4. Step each 90° bend in the system down in size (from sprue to gate) approximately 1.5mm (1/16") to reduce pressure drop. 5. Place vents at each 90° intersection and vent to atmosphere. 6. 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 1.5 times the diameter of the runner provide the 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 circular parts, full perimeter venting is recommended. 4. Cut vents depths to: 5. PC Compounds: 0.001"–0.002" depth and 0.250" width PC/PSU Compounds: 0.002"–0.003" depth and 0.250" width PES Compounds: 0.003"–0.004” depth and 0.250" width PEI Compounds: 0.001"–0.003” depth and 0.250" width PP Compounds: 0.001"–0.002” depth and 0.250" width ABS Compounds: 0.0015"–0.0025” depth and 0.250" width PEEK Compounds: 0.002"–0.004” depth and 0.250" width Nylon Compounds: 0.002" min. depth and 0.250" width Increase vent depth to 1.0mm (0.040") at 4.0mm (0.250") away from the cavity and vent to atmosphere.
Draft Angle	1. Maintain a minimum draft angle of 1/2° per side.
Startup & Shutdown	**Recommendations**
Purge Compound	HDPE or HIPS
Recycling	Recycling Gravi-Tech up to 25% is permissible. Testing the application is highly recommended to determine the effect recycling has on the desired physical properties. Recycling above 25% will have to submit to validation testing, before it can be accepted.

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
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

Troubleshooting Recommendations (continued)

Problem	Cause	Solution
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
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

Problem	Cause	Solution
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 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
Fibers on Surface (Splay)	Melt Temperature too low	1. Increase melt temperature 2. Increase mold temperature 3. Increase injection speed
	Insufficient Packaging	1. Increase pack and hold pressure, and time 2. Increase shot size 3. Increase gate size
Blush	Injection too quickly through gate	1. Reduce injection velocity 2. Increase/Decrease melt temperature 3. Increase/Decrease mold temperature
	Gate Size/Location	1. Increase gate size 2. Remove sharp corners from gate detail 3. Change gate location
Voids	Insufficient packing pressure	1. Increase hold pressure 2. Increase hold time 3. Increase/Decrease melt temperature 4. Increase mold temperature
	Impaired venting	1. Check vents for blockages 2. Insure proper cushion

For questions or issues, please contact
Specialty Engineered Materials Technical Support at 1.844.4AVIENT
www.avient.com