PROCESSING GUIDE

LubriOne™

INTERNALLY LUBRICATED FORMULATIONS

AVIENT™

LubriOne™

LubriOne™ Internally Lubricated Formulations are designed to be self-lubricating materials, offering low coefficient of friction and improved wear-resistance properties. These materials combine the unique benefits of internal lubricants such as PTFE, silicone and molybdenum disulfide with a wide array of base engineering resins. LubriOne materials have been demonstrated to reduce friction, noise, vibration, heat buildup, and improve product durability.

BASE RESIN	PPA	PC	PSU	PES	PPS	CO-POLYMER ACETAL	PEEK	PA
Barrel Temperatures* °F (°C)								
Rear Zone	550–580 (288–305)	520–560 (271–293)	600–640 (316–338)	630–660 (332–338)	550–580 (288–304)	350–370 (177–188)	660–700 (349–371)	440–490 (227–254)
Center Zone	560–600 (293–316)	530–570 (277–299)	620–670 (327–354)	650–680 (343–360)	560–615 (293–324)	380–390 (193–200)	700–730 (371–388)	470–510 (243–266)
Front Zone	580–620 (304–327)	550–580 (288–305)	630–680 (332–360)	670–730 (354–388)	590–630 (310–332)	390–430 (200–221)	720–750 (382–400)	490–540 (254–282)
Nozzle	575–615 (302–324)	550–600 (288–316)	630–680 (332–360)	680–700 (360–371)	600–625 (316–330)	380–415 (193–213)	720–750 (382–400)	520–570 (271–300)
Melt Temperature	575–615 (302–324)	560–600 (293–316)	625–675 (330–358)	650–710 (343–377)	600–625 (316–330)	370–410 (188–210)	670–740 (354–393)	520–570 (271–300)
Mold Temperature	250–300 (121–150)	175–240 (80–116)	190–300 (88–150)	225–325 (107–164)	250–325 (121–164)	150–225 (66–107)	290–375 (143–190)	150–200 (66–93)
Pack & Hold Pressure	50%–75% of Injection Pressure							
Injection Velocity in/s	1.0–3.0							
Back Pressure psi	50							
Screw Speed rpm	50–90							
Drying Parameters °F (°C)	6 hrs @ 175 (80)	4 hrs @ 250 (121)	4 hrs @ 275 (135)	4 hrs @ 300 (150)	4 hrs @ 250 (121)	2 hrs @ 200 (93)	3 hrs @ 300 (150)	4 hrs @ 180 (82)
Allowable Moisture %	< 0.05	< 0.02	< 0.02	< 0.04	< 0.02	0.15–0.20	< 0.02	0.10–0.20
Cushion in	0.125–0.250							
Screw Compression Ratio	2.5:1–3.5:1	2.0:1–2.5:1	2.5:1–3.5:1	2.5:1–3.5:1	2.5:1–3.5:1	2.5:1–3.5:1	2.5:1–3.5:1	2.5:1–3.5:1
Nozzle Type	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	Reverse Taper
Clamp Pressure	5–6 Tons/in^2 of projected area of cavities and runner system							

* Barrel temperatures should be elevated for compounds designed for electrical insulative properties.

STARTUP & SHUTDOWN	RECOMMENDATIONS
Purge Compound	HDPE or HIPS
Recycling	Recycling LubriOne up to 20% is permissible. Testing the application is highly recommended to determine the effect recycling has on the desired physical properties.

MOLD DESIGN	RECOMMENDATIONS
Gates	• Many different 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. • Gate diameters equivalent to 50% of the average wall thickness are recommended. • Land lengths of 0.020"–0.035" (0.50mm–0.90mm) are typically recommended.
Runners	• Full-round runners or a modified trapezoid runner are the best designs. Half-round runners are not recommended. • Only naturally balanced runner systems (“H” pattern) are recommended. • Runner diameters larger than 0.150" (3.8mm) and not exceeding 0.375" (9.5mm) are recommended. • Step each 90° bend in the system down in size (from sprue to gate) approximately 1/16" (1.5mm) to reduce pressure drop. • Place vents at each 90° intersection and vent to atmosphere. • Hot runner molds are acceptable and should be sized by the manufacturer.
Cold Slug Wells	• Place these wells at the base of the sprue to capture the cold material first emerging from the nozzle. • Place wells at every 90° bend in the runner system. • Well depths approximately 1.5 times the diameter of the runner provide the best results.
Venting	• Place vents at the end of fill and anywhere potential knit/weld lines will occur. • All vents need to be vented to atmosphere. • For circular parts, full perimeter venting is recommended. • Cut vent depths to: - PPA Compounds: 0.0015"–0.0025" depth and 0.250" width - PC Compounds: 0.002"–0.004" depth and 0.250" width - PSU Compounds: 0.003"–0.004" depth and 0.250" width - PES Compounds: 0.003"–0.004" depth and 0.250" width - PPS Compounds: 0.002"–0.003" depth and 0.250" width - Acetal Compounds: 0.0015" minimum depth and 0.250" width - PEEK Compounds: 0.002"–0.004" depth and 0.250" width - Nylon Compounds: 0.002" minimum depth and 0.250" width • Increase vent depth to 0.060" (1.5mm) at 0.250" (4.0mm) away from the cavity and vent to atmosphere.
Draft Angle	• Maintain a minimum draft angle of 1/2° per side.

TROUBLESHOOTING RECOMMENDATIONS

PROBLEM	CAUSE	SOLUTION
Incomplete Fill	Melt and/or mold temperature too cold	• Increase nozzle and barrel temperatures • Increase mold temperature • Increase injection speed • Increase pack and hold pressure • Increase nozzle tip diameter • Check thermocouples and heater bands
	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
	Shot Size	• Increase shot size • Increase cushion
Brittleness	Melt temperature too low	• Increase melt temperature • Increase injection speed • Measure melt temperature with pyrometer
	Degraded/Overheated material	• Decrease melt temperature • Decrease back pressure • Use smaller barrel/excessive residence time
	Gate location and/or size	• Relocate gate to nonstress area • Increase gate size to allow higher flow speed and lower molded-in stress
Fibers on Surface (Splay)	Melt temperature too low	• Increase melt temperature • Increase mold temperature • Increase injection speed
	Insufficient packing	• Increase pack and hold pressure, and time • Increase shot size • Increase gate size
Sink Marks	Part geometry too thick	• Reduce wall thickness • Reduce rib thickness
	Melt temperature too hot	• Decrease nozzle and barrel temperatures • Decrease mold temperature
	Insufficient material volume	• Increase shot size • Increase injection rate • Increase packing pressure • Increase gate size
Flash	Injection pressure too high	• Decrease injection pressure • Increase clamp pressure • Decrease injection speed • Increase transfer position
	Excess material volume	• Decrease pack pressure • Decrease shot size • Decrease injection speed
	Melt and/or mold temperature too hot	• Decrease nozzle and barrel temperatures • Decrease mold temperature • Decrease screw speed

TROUBLESHOOTING RECOMMENDATIONS

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

Note: These are general processing conditions. Please contact Avient for processing conditions specific to your formulations.

1.844.4AVIENT
www.avient.com