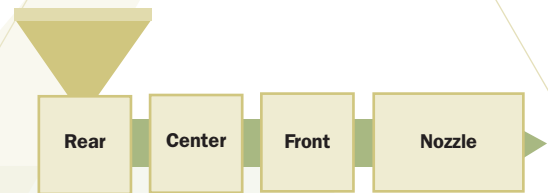




## LubriOne™ Wear Resistant Compounds

LubriOne™ Lubricated and Wear-Resistant Compounds have been specifically formulated 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™ compounds have been demonstrated to reduce friction, noise, vibration, heat buildup, and improve product durability.



### Injection Molding Parameters

Base Resin	PPA	PC	PSU	PES	PPS	Copolymer Acetal	PEEK	PA
<b>Barrel Temperatures* °C (°F)</b>								
<b>Rear Zone</b>	288 - 305 (550 - 580)	288 - 310 (550 - 590)	316 - 338 (600 - 640)	332 - 338 (630 - 660)	288 - 304 (550 - 580)	177 - 188 (350 - 370)	349 - 371 (660 - 700)	227 - 254 (440 - 490)
<b>Center Zone</b>	293 - 316 (560 - 600)	300 - 316 (570 - 600)	327 - 354 (620 - 670)	343 - 360 (650 - 680)	293 - 324 (560 - 615)	193 - 200 (380 - 390)	371 - 388 (700 - 730)	243 - 266 (470 - 510)
<b>Front Zone</b>	304 - 327 (580 - 620)	304 - 322 (580 - 630)	332 - 360 (630 - 680)	354 - 388 (670 - 730)	310 - 332 (590 - 630)	200 - 221 (390 - 430)	382 - 400 (720 - 750)	254 - 282 (490 - 540)
<b>Nozzle</b>	302 - 324 (575 - 615)	304 - 322 (580 - 630)	332 - 360 (630 - 680)	360 - 371 (680 - 700)	316 - 330 (600 - 625)	193 - 213 (380 - 415)	382 - 400 (720 - 750)	271 - 300 (520 - 570)
<b>Melt Temperature</b>	302 - 324 (575 - 615)	304 - 330 (580 - 625)	330 - 358 (625 - 675)	343 - 377 (650 - 710)	316 - 330 (600 - 625)	188 - 210 (370 - 410)	354 - 393 (670 - 740)	271 - 300 (520 - 570)
<b>Mold Temperature °C (°F)</b>	121 - 150 (250 - 300)	80 - 116 (175 - 240)	88 - 150 (190 - 300)	107 - 164 (225 - 325)	121 - 164 (250 - 325)	88 - 121 (190 - 250)	143 - 190 (290 - 375)	66 - 93 (150 - 200)
<b>Pack and Hold Pressure</b>	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure	50% - 75% of Injection Pressure
<b>Injection Velocity in/s</b>	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0	1.0 - 3.0
<b>Back Pressure psi</b>	50	50	50	50	50	50	50	50
<b>Screw Speed rpm</b>	50 - 90	50 - 90	50 - 90	50 - 90	50 - 90	50 - 90	50 - 90	50 - 90
<b>Drying Parameters °C (°F)</b>	6hrs @ 80 (175)	4hrs @ 121 (250)	4hrs @ 135 (275)	4hrs @ 121 (250)	3hrs @ 150 (300)	2hrs @ 93 (200)	3hrs @ 135 (275)	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	0.125 - 0.250	0.125 - 0.250	0.125 - 0.250
<b>Screw Compression Ratio</b>	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
<b>Nozzle Type</b>	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	General Purpose	Reverse Taper
<b>Clamp Pressure</b>	5 - 6 Tons/in <sup>2</sup> of projected area of cavities and runner system							

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

STARTUP & SHUTDOWN	RECOMMENDATIONS
<b>Purge Compound</b>	HDPE or HIPS
<b>Recycling</b>	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
<b>Gates</b>	<ol style="list-style-type: none"> <li>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.</li> <li>Gate diameters equivalent to 50% of the average wall thickness are recommended.</li> <li>Land lengths of 0.50mm – 0.90mm (0.020” – 0.035”) are typically recommended.</li> </ol>
<b>Runners</b>	<ol style="list-style-type: none"> <li>Full-round runners or a modified trapezoid runner are the best designs. Half-round runners are not recommended.</li> <li>Only naturally balanced runner systems (“H” pattern) are recommended.</li> <li>Runner diameters larger than 3.8mm (0.150”) and not exceeding 9.5mm (0.375”) are recommended.</li> <li>Step each 90° bend in the system down in size (from sprue to gate) approximately 1.5mm (1/16”) to reduce pressure drop.</li> <li>Place vents at each 90° intersection and vent to atmosphere.</li> <li>Hot runner molds are acceptable and should be sized by the manufacturer.</li> </ol>
<b>Cold Slug Wells</b>	<ol style="list-style-type: none"> <li>Place these wells at the base of the sprue to capture the cold material first emerging from the nozzle.</li> <li>Place wells at every 90° bend in the runner system.</li> <li>Well depths approximately 1.5 times the diameter of the runner provide the best results.</li> </ol>
<b>Venting</b>	<ol style="list-style-type: none"> <li>Place vents at the end of fill and anywhere potential knit/weld lines will occur.</li> <li>All vents need to be vented to atmosphere.</li> <li>For circular parts, full perimeter venting is recommended.</li> <li>Cut vent depths to: <ul style="list-style-type: none"> <li>PPA Compounds: 0.0015” – 0.0025” depth and 0.250” width</li> <li>PC Compounds: 0.002” – 0.004” depth and 0.250” width</li> <li>PSU Compounds: 0.003” – 0.004” depth and 0.250” width</li> <li>PES Compounds: 0.003” – 0.004” depth and 0.250” width</li> <li>PPS Compounds: 0.002” – 0.003” depth and 0.250” width</li> <li>Acetal Compounds: 0.0015” min. depth and 0.250” width</li> <li>PEEK Compounds: 0.002” – 0.004” depth and 0.250” width</li> <li>Nylon Compounds: 0.002” min. depth and 0.250” width</li> </ul> <p>Increase vent depth to 1.5mm (0.060”) at 4.0mm (0.250”) away from the cavity and vent to atmosphere.</p> </li> </ol>
<b>Draft Angle</b>	<ol style="list-style-type: none"> <li>Maintain a minimum draft angle of 1/2° per side.</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 on Surface (Splay)</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> <li>3. Increase gate 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