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.

<table>
<thead>
<tr>
<th>Base Resin</th>
<th>ABS</th>
<th>PA</th>
<th>PBT</th>
<th>PC</th>
<th>PE</th>
<th>PEEK</th>
<th>PP</th>
<th>PPS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barrel Temperatures °F (°C)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drying Parameters</td>
<td>190 (90) 2–4 Hours 0.01%–0.15%</td>
<td>180 (82) 4–5 Hours 0.10%–0.20%</td>
<td>275 (135) 3–4 Hours 0.02%–0.04%</td>
<td>250 (125) 3–4 Hours 0.02%</td>
<td>160 (70) 2 Hours 0.10%</td>
<td>300 (150) 3–4 Hours 0.10%</td>
<td>160 (70) 2 Hours 280 (135) 2–3 Hours 0.01%–0.20%</td>
<td></td>
</tr>
<tr>
<td>Nozzle Type</td>
<td>General Purpose</td>
<td>Nylon or Reverse Taper</td>
<td>General Purpose</td>
<td>General Purpose</td>
<td>General Purpose</td>
<td>General Purpose</td>
<td>General Purpose</td>
<td>General Purpose</td>
</tr>
<tr>
<td>Injection Velocity</td>
<td>2–5 in/sec; 50–127 mm/sec</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Injection Pressure</td>
<td>Medium to high (may range from 1,000–4,000 psi; 7–28 Mpa)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Back Pressure</td>
<td>50–100 psi; 0.3–0.7 Mpa</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Screw Speed</td>
<td>25–75 RPM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cushion</td>
<td>0.125–0.250 in; 3–6 mm</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Screw Compression Ratio</td>
<td>2.0:1–2.5:1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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.
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.

<table>
<thead>
<tr>
<th>Mold Design</th>
<th>Recommendations</th>
</tr>
</thead>
</table>
| 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

<table>
<thead>
<tr>
<th>Recommendations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Purge Compound</td>
</tr>
<tr>
<td>Recycling</td>
</tr>
</tbody>
</table>
# Troubleshooting Recommendations

<table>
<thead>
<tr>
<th>Problem</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
</table>
| 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 |
| Britleness                     | 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)

<table>
<thead>
<tr>
<th>Problem</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
</table>
| **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   
<pre><code>                      | 5. Move gate location   |
</code></pre>
<table>
<thead>
<tr>
<th>Problem</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
</table>
| 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 |