



**Maxxam<sup>™</sup>**  
FILLED AND REINFORCED  
POLYOLEFINS

# Maxxam™ Filled and Reinforced Polyolefins

The Maxxam™ portfolio of filled and reinforced polyolefins has been developed to meet the performance demands of various industry-specific applications. These formulations incorporate the latest in polymer modification, additive and process chemistry, and offer ease of processing in most standard thermoplastic equipment.

## INJECTION MOLDING PARAMETERS

	MINERAL REINFORCED PP & TPO	GLASS REINFORCED PP
<b>Barrel Temperatures*</b> °C (°F)		
Rear zone	204-215 (400-420)	213-224 (415-435)
Center zone	210-221 (410-430)	218-230 (425-445)
Front zone	215-227 (420-440)	224-235 (435-455)
Nozzle	212-224 (415-435)	221-232 (430-450)
<b>Mold Temperature</b> °C (°F)	16-50 (60-120)	16-50 (60-120)
<b>Pack Pressure</b>	50%-75% of injection pressure	50%-75% of injection pressure
<b>Hold Pressure</b>	50% of injection pressure	50% of injection pressure
<b>Injection Velocity</b> in/s	2.0-3.0	2.0-3.0
<b>Back Pressure</b> psi	50-100	50-100
<b>Screw Speed</b> rpm	30-100	30-100
<b>Drying Parameters</b> °C (°F)	Not required	Not required
<b>Cushion</b> in	0.125-0.250	0.125-0.250
<b>Screw Compression Ratio</b>	2.5:1-3.0:1	2.5:1-3.0:1
<b>Nozzle Type</b>	General purpose	General purpose
<b>Clamp Pressure</b>	2-4 Tons/in <sup>3</sup>	2-4 Tons/in <sup>3</sup>
<b>Screw Type</b>	General purpose polyolefin L/D ratio = 16:1-20:1	General purpose polyolefin L/D ratio = 16:1-20:1



STARTUP & SHUTDOWN	RECOMMENDATIONS
<b>Drying</b>	Drying is only recommended for visible surfaces. If required, drying recommended at 75°C (167°F) for 2 hours.
<b>Purge Compound</b>	Polypropylene
<b>Coloring</b>	Contact your PolyOne representative.
<b>Recycling</b>	Maxxam is fully recyclable. Conventional granulators with sharp blades should be used. Consistent regrind usage of up to 20% is permissible. Excessive fines or dust-like particles should be avoided.

MOLD DESIGN	RECOMMENDATIONS
<b>Tool Steel</b>	<ul style="list-style-type: none"> <li>• P20 tool steel is recommended for most Maxxam polyolefins.</li> <li>• H13 is recommended for glass-filled materials.</li> </ul>
<b>Gates</b>	<ul style="list-style-type: none"> <li>• All 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 should be equivalent to 50%-75% of the average wall thickness of the part to be injected.</li> <li>• A land length of 0.040" (1.0mm) is recommended.</li> </ul>
<b>Runners</b>	<ul style="list-style-type: none"> <li>• Full-round or modified trapezoid runners are the best design and provide the least surface to cross-section ratio. Half-round or standard trapezoid runners are not recommended.</li> <li>• Only naturally balanced runner systems ("H" pattern) are recommended.</li> <li>• Each 90° bend in the system should step down in size.</li> <li>• Vents should be placed at the intersection of each 90° bend off of the cold slug well and vented to atmosphere.</li> <li>• Hot runner molds are acceptable and should be sized by the manufacturer. Externally heated manifolds are recommended.</li> </ul>
<b>Cold Slug Wells</b>	<ul style="list-style-type: none"> <li>• Place cold slug wells at the base of the sprue to capture the cold material first emerging from the nozzle.</li> <li>• Place cold slug wells at every 90° bend in the runner system.</li> <li>• Well depths approximately 2-3 times the diameter of the runner provide best results.</li> </ul>
<b>Venting</b>	<ul 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 unreinforced polyolefin formulations, cut vent depths to 0.0005"-0.001" with a minimum 0.030" land length. Increase the vent depth to 0.010" at 0.100" away from the cavity and vent to atmosphere.</li> <li>• For reinforced polyolefin formulations (greater than 20% filler), cut vent depths to 0.001"-0.002" with a minimum 0.030" land length. Increase the vent depth to 0.010" at 0.100" away from the cavity and vent to atmosphere.</li> <li>• Vents should be placed at the intersection of each 90° bend in the runner system off of the cold slug well and vented to atmosphere.</li> </ul>
<b>Draft Angle</b>	<ul style="list-style-type: none"> <li>• Draft angle should be 1/2°-1° per side. Additional draft may be required for grained/textured surfaces.</li> </ul>

## TROUBLESHOOTING RECOMMENDATIONS

PROBLEM	CAUSE	SOLUTION
<b>Incomplete Fill</b>	Melt and/or mold too cold	<ul style="list-style-type: none"> <li>• Increase nozzle and barrel temperatures</li> <li>• Increase mold temperature</li> <li>• Increase injection rate</li> <li>• Increase pack and hold pressure</li> <li>• Increase nozzle tip diameter</li> <li>• Check thermocouples and heater bands</li> </ul>
	Mold design	<ul style="list-style-type: none"> <li>• Enlarge or widen vents and increase number of vents</li> <li>• Check that vents are unplugged</li> <li>• Check that gates are unplugged</li> <li>• Enlarge gates and/or runners</li> <li>• Perform short shots to determine fill pattern and verify proper vent location</li> <li>• Increase wall thickness to move gas trap to parting line</li> </ul>
	Shot Size	<ul style="list-style-type: none"> <li>• Increase shot size</li> </ul>
<b>Brittleness</b>	Low melt temperature	<ul style="list-style-type: none"> <li>• Increase melt temperature</li> <li>• Increase injection rate</li> <li>• Measure melt temperature with pyrometer</li> </ul>
	Degraded/Overheated material	<ul style="list-style-type: none"> <li>• Decrease melt temperature</li> <li>• Decrease back pressure</li> <li>• Use smaller barrel/excessive residence time</li> </ul>
	Gate location and/or size	<ul style="list-style-type: none"> <li>• Relocate gate to nonstress area</li> <li>• Increase gate size to allow higher flow rate and lower molded-in stress</li> </ul>
<b>Fibers/Minerals on Surface or Uneven Surface Appearance</b>	Melt temperature too low	<ul style="list-style-type: none"> <li>• Increase melt temperature</li> <li>• Increase mold temperature</li> <li>• Increase injection speed</li> </ul>
	Insufficient packing	<ul style="list-style-type: none"> <li>• Increase pack and hold pressure, and time</li> <li>• Increase shot size</li> </ul>
<b>Sink Marks</b>	Part geometry too thick	<ul style="list-style-type: none"> <li>• Reduce wall thickness</li> <li>• Reduce rib thickness</li> </ul>
	Melt too hot	<ul style="list-style-type: none"> <li>• Decrease nozzle and barrel temperatures</li> <li>• Decrease mold temperature</li> </ul>
	Insufficient material volume	<ul style="list-style-type: none"> <li>• Increase shot size</li> <li>• Increase injection rate</li> <li>• Increase packing pressure</li> <li>• Increase gate size</li> </ul>
<b>Flash</b>	Injection pressure too high	<ul style="list-style-type: none"> <li>• Increase clamp pressure</li> <li>• Decrease injection rate</li> <li>• Increase transfer position</li> </ul>
	Excess material volume	<ul style="list-style-type: none"> <li>• Decrease pack pressure</li> <li>• Decrease shot size</li> <li>• Decrease injection rate</li> </ul>
	Melt and/or mold too hot	<ul style="list-style-type: none"> <li>• Decrease nozzle and barrel temperatures</li> <li>• Decrease mold temperature</li> <li>• Decrease screw speed</li> </ul>

## TROUBLESHOOTING RECOMMENDATIONS

PROBLEM	CAUSE	SOLUTION
<b>Color Streaks</b>	Incomplete color dispersion	<ul style="list-style-type: none"> <li>• Increase back pressure</li> <li>• Increase screw speed (RPM)</li> <li>• Verify color concentrate compatibility</li> <li>• Reduce rear zone temperature</li> <li>• Increase injection rate</li> </ul>
<b>Excessive Shrink</b>	Too much orientation	<ul style="list-style-type: none"> <li>• Increase packing time and pressure</li> <li>• Increase hold pressure</li> <li>• Decrease melt temperature</li> <li>• Decrease mold temperature</li> <li>• Decrease injection speed</li> <li>• Decrease screw rpm</li> <li>• Increase venting</li> <li>• Increase cooling time</li> </ul>
<b>Not Enough Shrink</b>	Too little orientation	<ul style="list-style-type: none"> <li>• Decrease packing pressure and time</li> <li>• Decrease hold pressure</li> <li>• Increase melt temperature</li> <li>• Increase mold temperature</li> <li>• Increase injection speed</li> <li>• Increase screw rpm</li> <li>• Decrease cooling time</li> </ul>
<b>Burning</b>	Melt and/or mold too hot	<ul style="list-style-type: none"> <li>• Decrease nozzle and barrel temperatures</li> <li>• Decrease mold temperature</li> <li>• Decrease injection rate</li> </ul>
	Mold design	<ul style="list-style-type: none"> <li>• Clean, widen and increase number of vents</li> <li>• Increase gate size or number of gates</li> </ul>
	Moisture	<ul style="list-style-type: none"> <li>• Verify material is dried at proper conditions</li> </ul>
<b>Nozzle Drool</b>	Nozzle temperature too hot	<ul style="list-style-type: none"> <li>• Decrease nozzle temperature</li> <li>• Decrease back pressure</li> <li>• Increase screw decompression</li> <li>• Verify material has been dried at proper conditions</li> </ul>
<b>Weld Lines</b>	Melt front temperatures are too low	<ul style="list-style-type: none"> <li>• Increase pack and hold pressure</li> <li>• Increase melt temperature</li> <li>• Increase vent width and locations</li> <li>• Increase injection rate</li> <li>• Increase mold temperature</li> </ul>
	Mold design	<ul style="list-style-type: none"> <li>• Increase injection rate</li> <li>• Increase gate size</li> <li>• Perform short shots to determine fill pattern and verify proper vent location</li> <li>• Add vents and/or false ejector pin</li> <li>• Move gate location</li> </ul>
<b>Warp</b>	Excessive orientation	<ul style="list-style-type: none"> <li>• Increase cooling time</li> <li>• Increase melt temperature</li> <li>• Decrease injection rate</li> </ul>
	Mold design	<ul style="list-style-type: none"> <li>• Increase number of gates</li> </ul>
<b>Sticking in Mold</b>	Cavities are overpacked	<ul style="list-style-type: none"> <li>• Decrease injection rate</li> <li>• Decrease pack and hold pressure</li> <li>• Decrease nozzle and barrel temperatures</li> <li>• Decrease mold temperature</li> <li>• Increase cooling time</li> </ul>
	Mold design	<ul style="list-style-type: none"> <li>• Increase draft angle</li> </ul>
	Part is too hot	<ul style="list-style-type: none"> <li>• Decrease nozzle and barrel temperatures</li> <li>• Decrease mold temperature</li> <li>• Increase cooling time</li> </ul>

1.866.POLYONE  
[www.polyone.com](http://www.polyone.com)



Copyright © 2018, PolyOne Corporation. PolyOne makes no representations, guarantees, or warranties of any kind with respect to the information contained in this document about its accuracy, suitability for particular applications, or the results obtained or obtainable using the information. Some of the information arises from laboratory work with small-scale equipment which may not provide a reliable indication of performance or properties obtained or obtainable on larger-scale equipment. Values reported as "typical" or stated without a range do not state minimum or maximum properties; consult your sales representative for property ranges and min/max specifications. Processing conditions can cause material properties to shift from the values stated in the information. PolyOne makes no warranties or guarantees respecting suitability of either PolyOne's products or the information for your process or end-use application. You have the responsibility to conduct full-scale end-product performance testing to determine suitability in your application, and you assume all risk and liability arising from your use of the information and/or use or handling of any product. POLYONE MAKES NO WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, either with respect to the information or products reflected by the information. This literature shall NOT operate as permission, recommendation, or inducement to practice any patented invention without permission of the patent owner.