OnFlex™ V Series Thermoplastic Elastomer

Injection Molding Processing Sheet

OnFlex™ V Series Thermoplastic Elastomers are based on Thermoplastic Vulcanizate (TPV) technology that delivers the performance of a rubber in a thermoplastic material based upon polyolefin resins and cross-linked EPDM. These products are molded on standard thermoplastic equipment but exhibit key rubber-like properties such as compression set resistance, chemical resistance and a higher operating temperature limit than most thermoplastic elastomers. OnFlex V Series TPEs are injection-molded and overmolded to rigid polypropylene-based substrates with chemical adhesion. They are offered in both black and natural grades.

Injection Molding Parameters

OnFlex V TPEs are very forgiving materials to process and are not overly temperature-sensitive. The following temperatures should be used as a reference point and can vary ±5°C (±10°F). These temperatures should be used as a starting condition and can be increased to a maximum +10°C (+20°F).

Material	Nozzle °C (°F)	Front °C (°F)	Center °C (°F)	Rear °C (°F)
35-45A	195 (380)	190 (370)	190 (370)	175 (350)
50-55A	200 (390)	195 (380)	195 (380)	180 (360)
60-65A	200 (390)	195 (380)	195 (380)	180 (360)
70-75A	205 (400)	200 (390)	195 (380)	180 (360)
80-85A	205 (400)	200 (390)	195 (380)	190 (370)
40-60D	210 (410)	205 (400)	200 (390)	195 (380)

Successful injection molding of OnFlex V Series TPEs requires fast injection speeds to promote shear throughout the material, allowing it to flow. Temperature alone has minimal impact on the material's ability to flow. Typical polyolefin equipment should be used.

Processing & Equipment	Recommendations
Mold Temperature	15°C – 50°C (60°F – 120°F)
Screw Speed	50 – 150 rpm
Back Pressure	4 – 10 bar (50 – 150psi)
Injection Time	
First Stage	0.5 – 2.5 seconds
Second Stage	2.0 – 10 seconds
Pack Pressure	50% – 75% of injection pressure
Hold Pressure	50% of injection pressure
Cooling Time	10 – 40 seconds
Cushion	3mm – 6mm (0.125" – 0.250")
Screw Type	General-purpose polyolefin screw L/D ratio 16:1 – 20:1 Compression ratio 2.5:1 – 3.5:1
Nozzle	General purpose

STARTUP & SHUTDOWN	RECOMMENDATIONS
Drying	Drying is recommended for 3 hours at 82°C (180°F) with a desiccant drying system.
Purge Compound	Polypropylene
Coloring	Polypropylene color carriers should be used. Tumble blending at 3% – 5% color is recommended.
Recycling	OnFlex V TPEs are fully recyclable. Conventional granulators with sharp blades should be used. Consistent regrind usage up to 20% is permissible except for FDA- and NSF-regulated applications.

Small runners and gates should be used in the mold design for processing OnFlex V products. Due to the fast injection speeds, venting both the runners and the cavity is important to evacuate all gases from the mold. Gates should be placed in the thickest section of the part where the material will flow from thick to thin.

MOLD DESIGN	RECOMMENDATIONS
Gates	1. 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. 2. Gate diameters equivalent to 50% of the average wall thickness are recommended. 3. A short land length of 0.25mm – 0.50mm (0.010" – 0.020") is recommended.
Runners	1. Design runners as small as possible to provide shear heating and fast cycle times. Full-round runners or modified trapezoid runners are the best designs. Half-round runners are not recommended. 2. Only naturally balanced runner systems (“H” pattern) are recommended. 3. Runner diameters greater than 2.0mm (0.080") and not exceeding 6.0mm (0.250") are recommended. 4. Reduce each 90° bend in the system (from sprue to gate) approximately 1.5mm (1/16") to reduce pressure drop and promote shear. 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 vent depths to 0.020mm – 0.025mm (0.0008" – 0.001") and a land length of 4.0mm (0.160"). Increase vent depth to 1.5mm (0.060") at 4.0mm (0.160") away from the cavity and vent to atmosphere.
Draft Angle	1. Maintain a minimum draft angle of 1/2° per side.
Texturing	1. An EDM or sandblast finish is recommended. 2. A polished mold surface is not recommended, as the parts may tend to stick.
Overmolding	1. Minimum wall thickness of overmold should be 1.5mm (0.060"). 2. Shutoff for polypropylene-based materials: 0.08mm – 0.13mm (0.003" – 0.005").

TROUBLESHOOTING Problem	RECOMMENDATIONS 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.
	Mold design	1. Enlarge or widen vents and increase number of vents. 2. Check that vents are unplugged. 3. Add vacuum assist. 4. Check that gates are unplugged. 5. Enlarge gates and/or runners. 6. Perform short shots to determine fill pattern and verify proper vent location. 7. Increase wall thickness to move gas trap to parting line.
	Shot size	1. Increase shot size. 2. Increase cushion.
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.
Flash	Injection pressure too high	1. Decrease injection pressure. 2. Increase clamp pressure. 3. Decrease injection rate.
	Excess material volume	1. Decrease pack pressure. 2. Reduce 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.
Excessive Shrink	Too much orientation	1. Increase packing time and pressure. 2. Increase hold pressure. 3. Increase melt temperature. 4. Reduce mold temperature. 5. Reduce injection speed. 6. Reduce screw rpm. 7. Increase venting. 8. Increase cooling time.
Not Enough Shrink	Too little orientation	1. Decrease packing pressure. 2. Decrease hold pressure. 3. Decrease melt temperature. 4. Increase mold temperature. 5. Increase injection speed. 6. Increase screw rpm. 7. Decrease cooling time.
Splay at Gate	Moisture	1. Dry material. 2. Verify using desiccant drying system and virgin material.
Color Streaks	Incomplete color dispersion	1. Increase back pressure. 2. Use polypropylene carrier. 3. Reduce rear zone temperature. 4. Increase injection rate.

TROUBLESHOOTING Problem	RECOMMENDATIONS Cause	Solution
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. Use vacuum assist.
	Mold design	1. Reduce injection rate. 2. Increase gate size. 3. Perform short shots to determine fill pattern and verify proper vent location. 4. Move gate location.
Sticking in Mold	Cavities are overpacked	1. Reduce pack and hold pressure. 2. Reduce nozzle and barrel temperatures. 3. Reduce mold temperature. 4. Increase cooling time.
	Mold design	1. Increase texture on mold. 2. Increase draft angle.
	Part is too hot	1. Decrease nozzle and barrel temperatures. 2. Decrease mold temperature.

OnFlex V will exhibit post-mold shrinkage much like many thermoplastic elastomers. Many factors contribute to shrinkage such as processing conditions, part thickness, mold design and material grade. This data should be used as a base guideline.

SHRINKAGE Material	RECOMMENDATIONS Thickness	Shrinkage (in./in.)
35-45A	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.032 – 0.037 0.025 – 0.030 0.030 – 0.034
50-55A	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.029 – 0.033 0.020 – 0.024 0.023 – 0.027
60-65A	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.025 – 0.029 0.018 – 0.023 0.023 – 0.026
70-75A	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.022 – 0.025 0.017 – 0.021 0.022 – 0.025
80-85A	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.014 – 0.018 0.015 – 0.019 0.020 – 0.023
40-60D	1.9mm (0.074") 3.1mm (0.125") 5.0mm (0.197")	0.012 – 0.015 0.013 – 0.017 0.015 – 0.018

*** The data is based on 3" x 6" plaques with a melt temperature of 205°C (400°F) and a mold temperature of 27°C (80°F).**

PolyOne Corporation
33587 Walker Road
Avon Lake, OH 44012
1.866.POLYONE (1.866.765.9663)
www.PolyOne.com

PolyOne

At PolyOne, we deliver premium-quality products and services, which our customers use to enhance their own products. We believe no competitor can match our technical expertise because no workforce can outperform the people of PolyOne. Our people are ONE team working together – our strength and our pride. Their skills, passion and dedication are shaping a potent industry leader, a concerned corporate citizen and a great place to work.

OFVIMG-001-71256-B