

Top 10 Injection Mold Processing Tips for MDE TPV

- Completely purge all PVC or acetal resin from the machine before or after running our materials.
- Cooling time is dependent upon wall thickness, grade and tool design. Please consult our Injection Molding Guide.
- Cushion size is typically 3 to 6 mm (0.125 to 0.250").
- Hold time is typically 3 to 4 seconds and hold pressure is approximately 50% of injection pressure.
- It is far more effective to increase shear rate than to raise melt temperature for filling a mold.
- Injection rate should be 10 to 50 g/sec/gate.
- Injection speed should be medium (50%) to maximum (100%) of machine capacity.
- Injection times range from 0.5 to 2.5 seconds in order to optimize shear.
- Regrind can be used. Consistent regrind ratios are recommended to minimize process variations.
- Drying the material is recommended at 82 °C (180 °F) -- 3 hours for virgin material and 4 hours for reground material.

Top 10 Mold Design Tips for MDE TPV

- Draft angle of 0.50° per side is recommended.
- P-20 steel for core and cavity construction is recommended.
- Parts with undercuts can be ejected from the tool with conventional techniques.
- Design for a flow ratio of about 200, where flow ratio = (maximum flow length)/(average wall thickness). If flow ratio exceeds 200, include an additional gate to decrease flow length or increase wall thickness.
- A naturally balanced mold layout is optimum. Herringbone is NOT recommended.
- Shrinkage rates will vary with grade, wall thickness and part design. Please consult the Shrinkage TCD.
- Gate size diameter typically equals 30% - 50% of wall thickness.
- SPI/SPE #3 finish is required. Do NOT polish.
- Vent depth should be 0.04 mm (0.0015") maximum.
- Runners and gates are smaller in size than typical thermoplastic design.