This tip sheet is intended to provide suggested starting conditions for injection molding of HDPE resins but is not intended to be an all-inclusive guide. Many factors such as part design / mold design, venting and gate types, including location have a large influence on part quality, processing conditions can have the largest effect on part quality.

**Suggested Starting Conditions:**

<table>
<thead>
<tr>
<th>Nominal Melt Flow Rate</th>
<th>2 - 6</th>
<th>6 – 11</th>
<th>11 - 25</th>
<th>25 - 45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Melt Temperature</td>
<td>F</td>
<td>430 +/- 10</td>
<td>420 +/- 10</td>
<td>410 +/- 10</td>
</tr>
<tr>
<td>Rear</td>
<td>F</td>
<td>350 +/- 10</td>
<td>350 +/- 10</td>
<td>350 +/- 10</td>
</tr>
<tr>
<td>Middle</td>
<td>F</td>
<td>400 +/- 10</td>
<td>380 +/- 10</td>
<td>380 +/- 10</td>
</tr>
<tr>
<td>Front</td>
<td>F</td>
<td>410 +/- 10</td>
<td>390 +/- 10</td>
<td>390 +/- 10</td>
</tr>
<tr>
<td>Nozzle</td>
<td>F</td>
<td>420 +/- 10</td>
<td>400 +/- 10</td>
<td>400 +/- 10</td>
</tr>
<tr>
<td>Mold</td>
<td>F</td>
<td>50 - 70</td>
<td>50 - 70</td>
<td>50 - 70</td>
</tr>
</tbody>
</table>

Injection Pressure, 1st Stage

- 800 – 1,500
- 800 – 1,500
- 800 – 1,500
- 800 – 1,500

It is expected that these conditions will be systematically adjusted to achieve the best overall balance between cycle time, part quality and part performance.

**General Process Information:**

**Equipment**
High density PE can be molded in standard single-stage screw molding equipment without alterations. Pre-drying is not necessary with HDPE resins.

**Melt temperature**
Best results are obtained when HDPE is molded at temperatures ranging from 400 F up to 430 F. Too high of a temperature can cause problems with excessive flashing and burning and with shrink phenomena such as sink-marks, warpage, shrinkage, and void formation. Brittle parts also can be caused by either too high or too low of a temperature. Too low of a temperature can promote flow marks, weld lines, poor surfaces, lamination, short shots and undesirable molded-in stresses.

**Injection pressure**
The proper injection pressure depends largely on part size and configuration. Pressures usually range from 800 to 1,500 psi. 1st-stage pressure should be high enough to fill ~99% of the part and to avoid problems with shrinkage, voids, sinks, and short-shots. Too much pressure can cause parts to flash, burn, and stick in the mold or warp.

**Injection time**
Injection time could, in some cases, take up a good portion of the overall cycle. Injection time plays a relatively minor role in controlling warpage as compared to its major role in the managing or controlling shrinkage.

**Mold temperature**
Mold temperatures usually range from 50-70 F and should be high enough to produce good part surfaces, and minimize molded-in stresses. Temperatures should not be so high however, that shrinkage, warpage, sinking, and cavity or core sticking become problems.
Tips for Injection Molding
INEOS O&P HDPE Resins

Hold time
Allow sufficient hold time to cool the part before removing it from the mold. Shortening the hold time can lead to increases in warpage, sinking marks, ejector pin scars and shrinkage.

Back Pressure
Using minimal back pressure, in the range of 50-100 psi (gauge) is conducive to improved cycle times. Higher back pressures may be used, however, if more screw shear for melting or pigment mixing is needed.

Mold release
Mold release agents are generally not necessary due to the excellent release characteristics of HDPE. Sticking problems that cannot be resolved by modifications to processing conditions can sometimes be corrected by minor mold changes.