

Restricted by company

# Shape Memory Polymer (SMP) Guide for Injection/Extrusion molding

Technical Note

June 20, 2014



## Technical Guide for Injection/Extrusion Molding of Shape Memory Polymer for MM type(pellet)

Injection mold is most popular mass production method. Injection molding provides good finish surface and accurate dimension as well as producing a desired shape. Shape Memory Polymer (SMP) is a sensitive material to temperature. Especially its viscosity is very sensitive to temperature. SMP shows good fluidability in a mold, so it does not require high pressure for injection. Following are our recommendation. Please refer.

### 1. Injection /Extrusion Molding Machine

It is recommended to select an injection machine, which has a capability having short material residence in a barrel not to degrade the material while is in there.

- a) Injection Volume: Injection volume should be 1.5 times larger than that of production volume.
- b) Screw L/D Ratio: (Flight length of screw)/(outside diameter of screw) =L/D is recommended to be larger than 18.
- c) Compression Ratio ; (Depth of feed section) / (Depth of metering section) is recommended to have 2 to 3.5

### 2. Mold by Injection Molding

As well as thermoplastic Polyurethane, similar care may be necessary.

#### a) Sprue

Compared to ordinary plastic like PP or ABS, a shorter Sprue bushing design is recommended. The preferable taper angle of it is 3 to 5 degrees.

Because of low softening temperature of (SMP), the sprue demold risk is exist. So it is recommended to take detachable sprue for a modification.

#### b) Runner

If full round runner is taken, it's diameter is recommended to be  $\Phi 5 \sim 8$  mm. Box designed runner is also feasible, in that case a larger draft taper is recommended.

#### c) Cold slug well

Use of cold slug well is not always necessary. However in case of cold runner design, cold slug well at the end of runner or sprue is effective to make a product surface smooth.

#### d) Gate

The desirable gate location is in thicker portion of a mold for a purpose of maintaining the holding pressure.

When the temperature of the melt polymer flow in the mold is low, a viscosity of the material is increased. So in the case of a mold product having a

thickness less than 1-mm, it is recommended to increase the number of gates. For your reference, flow length of it is estimated about 70 to 80 mm. A gate sectional area of 1.5 to 2.0 larger compared to ordinary polymer is preferable in order to reduce bubbles, voids.

Basically any gate type used for popular plastics can be applied to (SMP).

**e) Cavity**

Due to the shape memory character of (SMP), the deformations created during the ejection process tend to be memorized. When it is required to eject at the temperature above glass transition temperature of the material ( $T_g$ ), number of ejection pins having more than 1.5 times for ordinary polymer as well as larger draft angle of the cavity are recommended

**f) Venting**

The depth of a gap for venting shall be 0.01 to 0.03mm. The existence of the insert may alleviate trapped air problem.

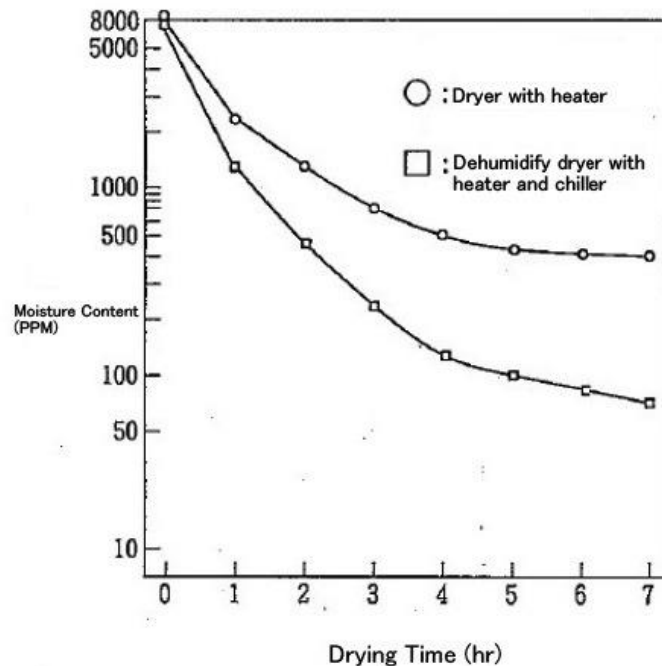
**g) Cooling**

Large cooling capacity is required, because the ejecting product is cooled down to the temperature lower than the  $T_g$ . Especially cooling capacity around the down stream position of sprue having a large diameter is desired to be increased

**3. Dehumidification of raw material (pellet)**

Please keep below 300ppm moisture concentration by drying.

It is recommended at 80 C X 4 Hours by dehumidify dryer (with chiller).



SMP MM pellet drying characteristics at 80°C in the air

#### 4. Suggestion for injection /Extrusion mold

##### a) Shrinkage

Mold shrinkage of the products after injection is estimated about 0.4 to 0.6%. In order to shorten an injection cycle, large shrinkage of several percent may result in, when a product is ejected out before its core is cool down at the temperature lower than the T<sub>g</sub> of a material.

In summer if the room temperature is above the T<sub>g</sub>, due to the release of residual stresses, large heat shrinkage also may occur.

##### b) Melt temperature

The desirable melt temperature of SMP for injection mold is 205 to 215 degrees C. Smaller shrinkage will be obtained with higher melt temperature for a product of thinner than 2 mm. In case of thicker than 5mm, lower melt temperature is better to avoid an appearance of void and babbles.

The recommended temperature profiles of injection machine smaller than 10 oz (about 284g) are as follows

Feed section : 160 to 180 °C

1st transition section and metering section: 190 to 210 °C

Nozzle : 195 to 205 °C

##### c) Injection speed

Higher injection speed should be selected for thin product such as 2 mm thick less, that is easy to get higher cooling rate and then the filling material is rapidly cooled. If there is surface defect such as silver streak in the product, slower speed is required. If it is not right remedy for short shot or surface defect, it is recommended to set the barrel temperature lower by 5-10 degree Celsius.

##### d) Holding time

1.5 to 2 times longer holding time is recommended compared to ordinary plastic.

##### e) Screw back pressure

For the purpose of air vent among pellets, about 5Mpa is required. However the screw back pressure should not exceed 10Mpa

##### f) Surface temperature

The surface temperature of the mold is preferably adjusted at 5 to 10 °C lower than the T<sub>g</sub> temperature of Shape memory polymer (SMP). In case of utilizing the material of which T<sub>g</sub> temperature is lower than 35 °C, the

surface temperature of the mold should be lower than 20 °C. In this case, some care must be taken to avoid dew on the cavity surface.

**Troubleshooting Guide and Remedy**

Trouble	Remedy
short shot	Increase melt temperature Increase the temperature in the barrel about 5 °C in each sections in order
	In case of cooling tool is used, the temperature to be set in the machine shall be lower than Tg by 5 to 10 °C
	Increase injection speed
	or Increase boosting pressure
	In case of short shot due to the lack of gas ventilation, decrease injection speed.
Voids	Increase holding time
	Increase holding pressure
	Increase the mold temperature
	Decrease the temperature in the barrel
	In case of the rapid cooling rate for thin product, increase injection speed
Flow mark Surface imperfection Surface bloom	Increase melt temperature
	Increase back pressure
	Decrease rotation speed of screw
	Decrease injection speed for a part of Surface bloom
Silver streaking	Dry material to the extent of vapor contents is less than 300ppm In this case, usually dehumidified dry are is used
	Decrease melt temperature
	Reduce screw RPM
Silver streaking	Desirable back pressure is 5-10Mpa
	Reduce nozzle temperature to avoid the drooling out of the nozzle.
	Reduce suck back stroke to be smaller than 5 mm
	Eliminate the water leak from mold and clean the mold surface
	In case of pin gate, reduce injection speed
Bubble	Dry material to extent of vapor contents is less than 300ppm In this case, usually dehumidified dry are is used
	Decrease melt temperature
	Increase holding pressure
	Increase holding time
	Increase injection speed