

Injection Molding Guide for KEPAMID and KEPEX

R&D Center

1. Injection molding machine

Injection molding is a manufacturing process for producing parts from both thermoplastic and thermosetting plastic materials. It is important to understand the process of injection molding in order to obtain high quality products.

One cycle of the injection molding process is simple. Material is first dried at the recommended drying condition and then it is fed into a heated barrel, mixed, and forced into a mold cavity where it cools and hardens to the configuration of the mold cavity.

More specific guidelines are:

- (1) The non-return valve or check ring should be regularly checked to achieve holding pressure and a proper amount of cushion, otherwise some issues such as sink marks, or variations of weight and dimension of the molded parts can occur.
- (2) An open nozzle is recommended for individual band heaters on the cylinder in order for separate heat control. It should be noted that it is very dangerous if the nozzle cools and the melted material hardens because the solid material will block the nozzle, which will cause a pressure increase inside the cylinder.
- (3) The compression zone in the screw is recommended to be in the range of 25~30 %. If the compression zone is too small, the melted material can be decomposed due to excessive shear heating.

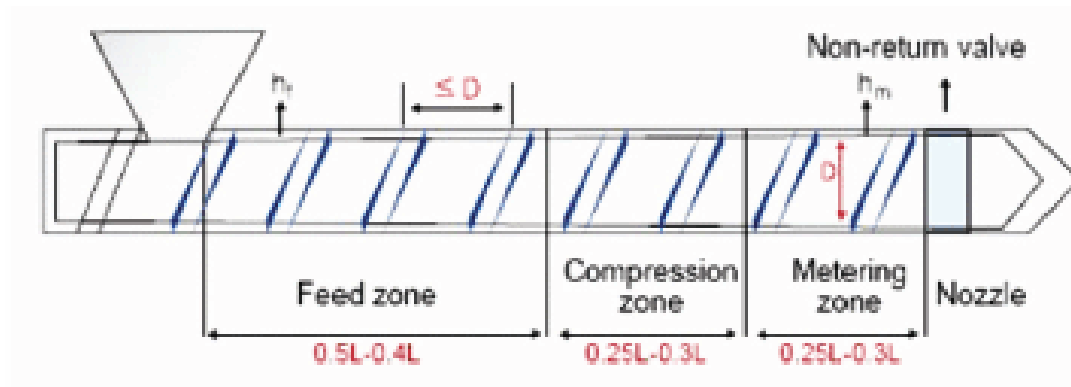


Figure 1. Typical injection molding screw of PA, PBT

<Recommendation for injection molding machine>

- 1) The one shot weight: 20 ~ 50 % of machine capacity
- 2) Screw diameter: Small or Medium
- 3) Compression ratio: 3/1 ~ 4/1
- 4) L/D: More than 20

2. Injection molding condition

In mold fabrication, it is essential to previously review the dimensional precision, flow characteristics of the raw material, consistency of the product, cost-effectiveness, etc.

(1) Pre-drying

Pre-drying of PA6 and PA66 is very important because they are hydrophilic materials, which can easily absorb water and cause a surface problem on the molded article. Sometimes, the injection molding process for these materials is difficult due to an excessive drooling problem. In addition, even though PBT and PET belong to polyester groups, which rarely absorb water, they can become degraded while processing due to hydrolysis reaction. When hydrolysis reaction occurs, it can cause deterioration of the material's tensile and impact strength and surface issues such as flashes or silver streaks on the molded articles. To minimize the problem caused by water absorption, the water content of the material's pellets should be maintained below 0.02 %. In particular, because PET can absorb water faster than PBT, it should be dried thoroughly for good quality of the molded articles.

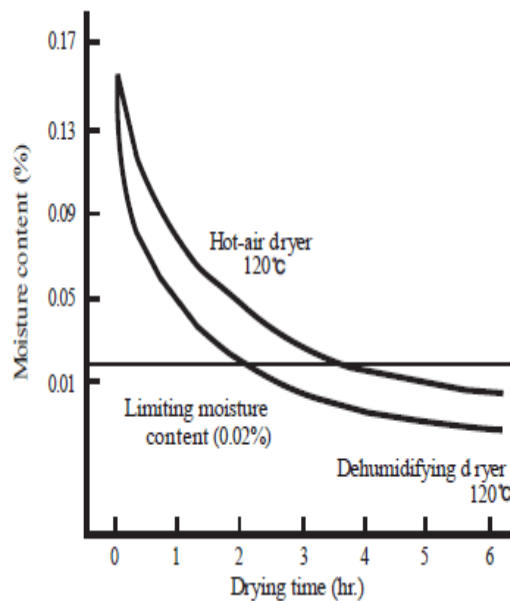


Figure 2. Water content of PET with drying time

Pre-drying conditions of PA6, PA66, PBT and PET are as follows.

Polymer	Dryer type	Temperature	Time	Water Content
PA6 PA66	Dehumidifying Hopper	80℃ 90℃	4-6h 6-8h	0.05%
PBT PET	Hopper	120-130℃	3-5h	0.02%

(2) Melt temperature

The melt temperature is generally higher than the set cylinder temperature due to shear heating of the screw rotation. The recommended cylinder temperatures are as follows:

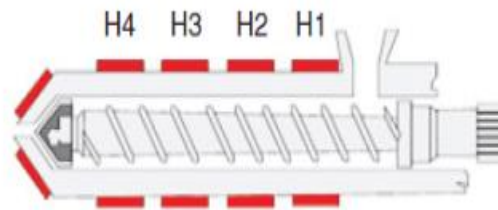


Figure 3. Cylinder temperature set

Polymer	GRADE	H4	H3	H2	H1
PA6	Unfilled	240	240	240	230
	Impact	250	250	250	240
	Glass fiber	260	250	250	250
	Mineral filler	250	240	245	230
PA66	Unfilled	275	275	275	270
	Impact	285	280	280	270
	Glass fiber	290	285	285	280
	Mineral filler	290	285	285	280
	Unfilled FR*	275	265	270	260
	GF, FR*	300	290	285	280
PBT	Unfilled	250	240	240	230
	Unfilled FR*	245	235	245	230
	Glass fiber	250	250	240	230
	GF, FR*	250	240	245	230
PET	Glass fiber	280	280	270	260
	GF, FR*	280	275	270	260

(3) Injection pressure

Injection pressure should be set high enough to achieve a high injection speed that may not be lowered by a low injection pressure. The appropriate injection pressure range is between 800 ~ 2000 bar.

(4) Mold temperature

The mold temperature is one of the most important parameters for injection molding of crystalline polymer in particular. The mold temperature may be widely set at 60 ~ 120°C, and a general recommendation is 70 ~ 90°C for general purpose use of KEPAMID® and KEPEX®.

If surface finish is important or the service temperature of a finished part is expected to be high, a higher mold temperature is recommended.

To obtain a good quality product, the mold temperature must be consistently maintained so that the temperature distribution in the mold may be achieved uniformly.

(5) Injection speed

The injection speed should be determined by part geometry, gate size and location, surface features, flow characteristics, mold temperature, etc. In general, injection speed is set to high when there are flow marks, record marks, and sink marks; on the other hand, a low injection rate is good to prevent jetting, flush, burn marks, or gate smears, which are generated by high shear force against the cavity wall.

(6) Hold pressure

Hold pressure plays a key role in optimizing parts in not only dimension, but also in mechanical and physical properties. Because in the hold stage (hold/pack), remaining melted material for about 1~5 % of a cavity is forced to fill into the cavity to compensate for the volume contraction during cooling. The hold (pack) time must be set to slightly exceed the gate seal time (normally $\frac{1}{2}$ to 1 sec) at which a gate is completely solidified so that a constant product may be obtained. As shown in Figure 4, the weight of a molded part increases upon the hold pressure time and then stops at a certain point.

At this time, the gate of the part is solidified entirely and no more material can be incorporated. Finally, the part weight becomes constant after the gate seal time.

It is recommended that the hold pressure time be maintained until the gate seal is completed. Because the gate sealing time changes mostly upon the shape of cross-section and mold temperature, a proper hold pressure time must be determined such that the weight and dimension of a molded product are within a certain range.

By setting optimum hold pressure, molded parts products with consistent dimensions can be produced. As a rule of thumb, the hold time can be simply calculated by multiplying wall thickness (mm) times 8.

The hold pressure must be set in consideration of dimensional requirements. As a rule, hold pressure amounts to between 60-90 % of the injection pressure.

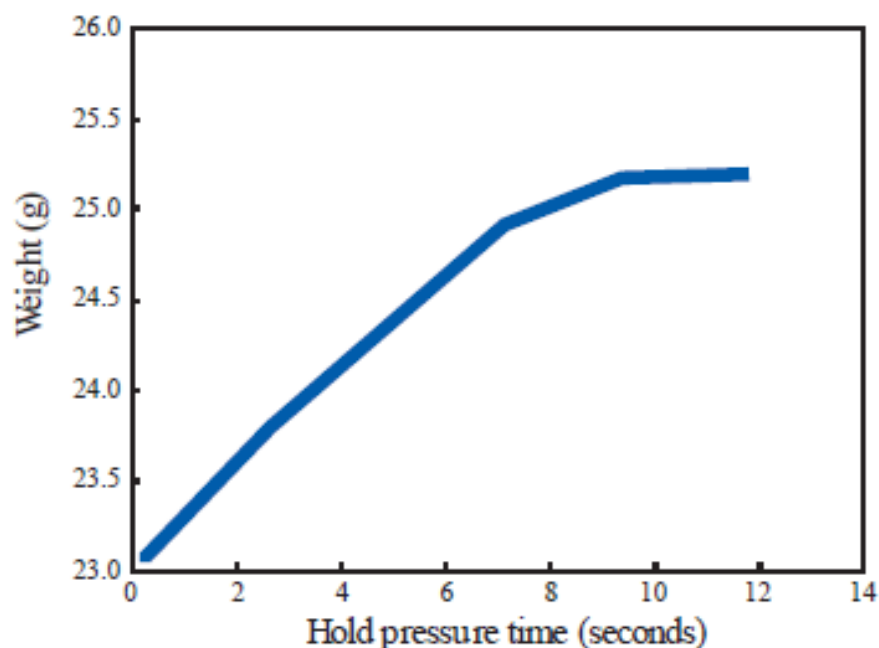


Figure 4. Hold pressure time and product weight

(7) Plasticizing

Because a fast screw speed can cause material to decompose by high shear force, the reciprocating speed is preferably set as low as possible unless it affects cycle time. Generally, 80 ~ 120 screw RPM is recommended. But because screw speed is dependent on screw diameter, screw speed should be decreased with larger screw diameters.

A back pressure of 50 ~ 100 bar (hydraulic pressure usually 5 ~ 10 bar) is generally appropriate. However, to increase the efficiency of the dispersion of a color master batch (color concentrates) or pigment, higher mixing by increasing back pressure may be required.

In addition, high back pressure may be used to eliminate un-melted particles. In the case of glass fiber-reinforced grades, high back pressure, proportional to rotational speed leads to breakage of the glass fiber, resulting in deterioration of mechanical strength. More importantly, excessive back pressure gives rise to lower output along with longer cycle times. Therefore, it should be taken into consideration when optimizing the back pressure.

(8) Cooling

The total cooling time is determined as the sum of “hold pressure time + screw retraction time + a shot safety margin”.

Once material is entirely solidified, no additional cooling time is needed. Most of the time affecting the cooling time is the hold time. Therefore, if hold pressure time is set appropriately, only screw retraction time needs to be taken into account.

| Calculation of theoretical cooling time |

$$S = \frac{t^2}{\pi^2 \alpha} \ln \left[\frac{8}{\pi^2} \frac{(T_c - T_m)}{(T_x - T_m)} \right] \quad \alpha = \frac{R}{C_p \rho}$$

S = Theoretical cooling time

t = Maximum part wall-thickness

α = Thermal diffusivity of material

R = Thermal conductivity

C_p = Specific heat

T_x = Ejection temperature of molding

T_m = Mold temperature

T_c = Cylinder temperature

3. Change material and interruption

(1) Changing material

In general, the cylinder has to be cleaned with a polyethylene or polypropylene both before and after processing. To prevent foreign material contamination and establish quality control, materials should be changed as little as possible.

(2) Interruption

If the molding cycle is stopped for a long time, the material in the cylinder can be decomposed and finally carbonized. This carbonized material is not easily separated from the screw. However, when the temperature is cooled down to room temperature, it contracts and can be separated from the screw, but imparts bad effects on the next molding. Therefore it is necessary, sometimes, to disassemble the screw and clean the carbonized material thoroughly.

HQ

Mapo-daero 119 (Gongdeok-dong) Hyeoseong Bldg.
Mapo-gu, Seoul, Korea
Tel 82-2-707-6840 ~ 8, Telefax 82-2-714-9235

KEP Americas

106 North Denton Tap Road Suite 210-202 Coppel,
TX 75019, USA
Tel +1 888 KEPITAL, Telefax +1 888 537-3291

KEP Europe GmbH

Rheingastrasse 190-196 D-65203 Wiesbaden, Germany
Tel +49 (0)611 962-7381, Telefax +49 (0)611 962-9132

KEP China

A1905, HongQiao Nanfeng Plaza, 100 Zunyi Road,
Shanghai, China
Tel +86 21 6237-1972, Telefax +86 21 6237-1803

Disclaimer: The information contained in this data sheet is based on our current knowledge and experience, so it may change as new knowledge and experience becomes available. This information is based on only above-mentioned product produced in Korea Engineering Plastics Co., Ltd. ("KEP") through relevant test methods and conditions and doesn't relate to any products made of this product with the inclusion of other additives, such as processing aids or colorants. This information should not be construed as a promise or guarantee of specific properties of this product described or its suitability for a particular application, so users make their own determination as to its suitability to their purposes prior to use this product. It is the sole responsibility of the users to investigate whether any existing patents are infringed by the use of this product. This product is not intended for use in medical and dental implants and users should meet all safety and health standards. KEP makes no warranty and assumes no liability in connection with any use of this information.