Technical Direction

Molding Guide

Iupital®

Acetal Copolymer ユピタール®

Mitsubishi Engineering-Plastics Corporation.

A joint corporation of Mitsubishi Gas Chemical Company, Inc. and Mitsubishi Chemical Corporation

Content

1. Injection machine

The most common injection machine for POM is inline-screw-type. The inline-screw has non-return ring (back-flow check valve) at the screw head, which will prevent melted resin leak back flow to cylinder.

1.1 Calculation of shot capacity required

Ideal shot capacity is shown as follows.

$$Q=(1.3\sim1.5)\times W$$

Q (g): Shot capacity of injection machine
W (g): resin weight for filling cavity including a sprue and runners

In case of machine capacity is not enough against molded parts volume, plasticizing of resin will be not enough, then non-melted resin will cause that the molded parts incidentally cannot get original properties because non-melted will inject into mold cavity.

On the other hand, when the machine capacity is excess against molded parts volume, retention time of resin in cylinder will be long and long retention time will be a cause of degradation of resin. So, normally from 30 to 80 % shot capacity of injection machine is used.

1.2 Calculation of clamping force required

Clamping system is able to use direct clamping mechanism, toggle clamping mechanism, or electric clamping mechanism for injection molding machine of Iupital. Internal average pressure in the cavity of Iupital injection is about 500 ~ 700 kg/cm^2, so clamping force is able to calculate by following equation.

$$P\geqq(0.5\sim0.7)\times A$$

P (ton): Clamping force of injection machine
A (cm^2): Projected area

1.3 Nozzle type of cylinder

Injection molding machine of Iupital usually use not only open nozzle but also can use shut off nozzle. Both nozzles should have temperature controller. It has better to use shut off nozzle to prevent stringing. However, when use shut off nozzle, it is necessary to keep attention because shut off nozzle will be a cause of silver streak or burnt by residual resin at dead space.

(1) Open nozzle (2) Shut off nozzle

Figure-1.1 Nozzle type of cylinder

1.4 Injection mechanism

Injection machine for Iupital requires basic performance injection machine such as one step injection velocity and two step holding pressure. If require more precise molding, should better to use multi step injection mechanism.

1.4.1 Multi program control

It is able to solve appearance problem, sink mark, warpage, flash, and dimensional stability of shot variation by controlling hydraulic pressure, screw position, screw rotation speed by using multi steps program control.

1.4.2 Multi step velocity control

The cause of appearance problem is a variation of flow front speed, so it is possible to take measures by controlling injection velocity.

In case of injection velocity is too low, then appears appearance problem, e.g. record groove, insufficient replication, weld line, or short shot. On the contrary, in case of injection velocity is too fast, also appears some problem, e.g. jetting, flow mark, air burn, or sink mark caused by insufficient air ventilation.

Figure-1.2 shows an example taking measures to avoid several molding defects by using multi steps program control.

In case of normal injection control such as without using multi steps program control, it has narrower good product range comparing using multi steps program control.

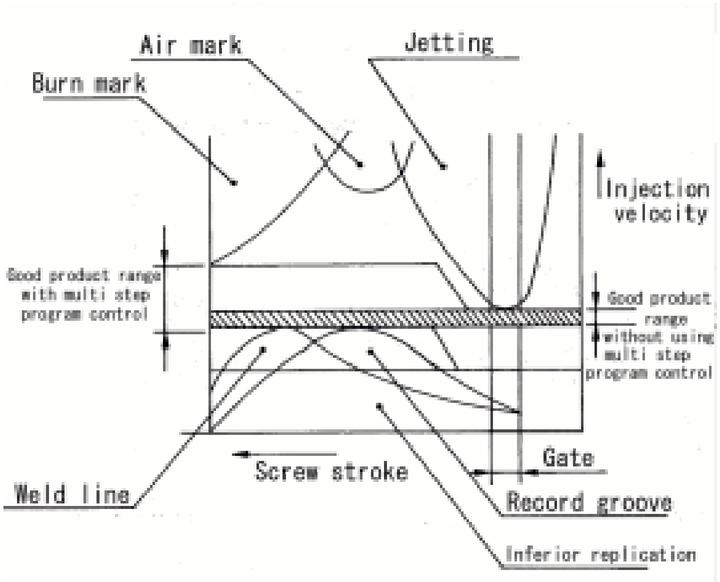


Figure-1.2

1.4.3 Holding pressure control

Molding defects such as sink mark, warpage, flash, etc. have relation to holding pressure condition. Therefore, it is able to take measures by controlling holding pressure condition.

Figure-1.3 shows holding pressure program pattern to avoid molding defects.

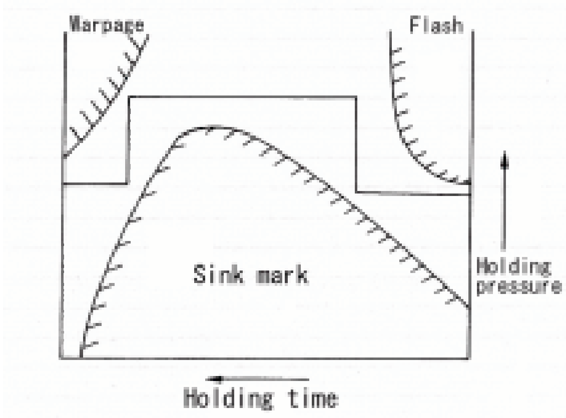


Figure-1.3

1.5 Non-return ring (back flow check valve)

Injection machine for Iupital is necessary to have a non-return ring (back-flow check valve) at screw head to avoid melt resin leak back to cylinder. If the non-return ring gets damage by abrasion or corrosion, back flow occurs during holding time. When back flow occurs, then the injection machine cannot keep cushion stroke and injection pressure cannot be effective to the cavity. If this case, it is difficult to get good molded parts. Therefore, it is necessary to pay attention to the stability of cushion stroke during injection molding. Figure-1.4 shows an example of non-return ring.

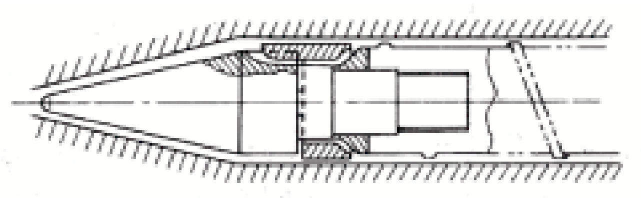


Figure-1.4 Non-return ring

2. Peripheral equipment

2.1 Material dryer

Water absorption of POM is small and normally packed into bag which is prevented moisture absorbing. Therefore, it is able to use without drying, when open the bag.

However, in case of to regard decreasing MD or getting good products as important, it is desirable to dry up at 80 ~ 90 degrees-C for 3 ~ 4 hours.

Hopper dryer or box circulating dryer can be used as material dryer for Iupital.

2.2 Mold temperature controller

Mold temperature controller is necessary to keep constant quality (dimension, physical properties etc.) in injection molding of Iupital. Mold temperature control systems are shown in **Table-2.1**.

Table-2.1 Mold temperature control systems

System	Adaptation for Polyacetal injection
Circulating hot-water type	Most popular system for POM injection. It is need to attention about scale of inside line. Use at mold temperature from 40 to 90℃
Pressurized circulating hot-water type	Usually use for POM It is need to attention about scale of inside line. Use at mold temperature from 40 to 150℃
Circulating hot-oil type	Use at mold temperature of ≧120℃
Electrical cartridge heater type	Unsuitable (impossible to avoid over-heating) Sometimes use for auxiliary system
Chiller	Effective for local temperature control e.g. over heating core

2.3 Exhaust system

Iupital has good thermal stability during normal processing, but thermal degradation can occur under unusual conditions. If the resin is over-heated, stimulative formaldehyde gas will generate. Formaldehyde gas has hard stimuli for eyes, nose, and throat. Therefore, exhaust system is recommended for molding factory of POM.

3. Molding conditions for Iupital

3.1 Pre-drying condition

Standard pre-drying condition of Iupital is normally at 80 degrees-C for 3 to 4 hours. Iupital is able to handle without drying. However, dried up Iupital resin is recommended to decrease mold deposit and to improve surface appearance.

Drying curve of POM resin is shown **Figure-3.1**.

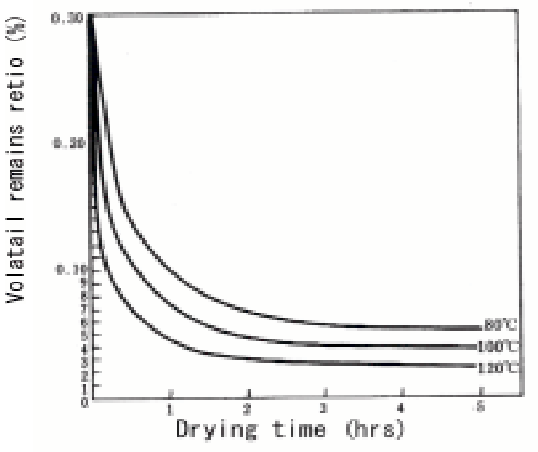


Figure-3.1 Drying curve of POM resin

3.2 Resin temperature

Standard molding conditions of Iupital various grade are shown in **Table-3.1**. Temperature dependence of POM flowability is quite small. If the cylinder temperature increases, flowability of POM will not so increase. Conversely, increasing resin temperature causes mold deposit and silver streak. On the other hand, too low resin temperature will cause non- melt resin flow. It is possible to lead insufficient mechanical properties. Therefore, resin temperature of POM is required at least 160 ~ 170 degrees-C.

Table-3.1 Standard molding condition of "Iupital" various grade: unit degrees-C

Grade	Nozzle	Nozzle side	Cylinder middle	Popper side
Standard (F10, F20, F30, F40)	１８０ ～ ２１０	１９０	１８０	１７０
Weather resistant		１９０	１８０	１７０
FG, MF, FB		２００	１９０	１８０
FC		２００	１９０	１８０
FT		１９０	１８０	１７０
FL, FW		１９０	１８０	１７０
LO, FX		１９０	１８０	１７０
FA		１９０	１７５	１７０
FS		１９０	１８０	１７０
FU		１９０	１８０	１７０
ET		１９０	１８０	１７０
TC		１９０	１８０	１７０

FS: Slow rotation speed of screw FU: Mold temperature of ≦40℃ ET: Slow injection speed

Setting maximum temperature: 220℃ Setting minimum temperature: 160℃

3.3 Injection pressure

Injection pressure consists of filling pressure and holding pressure. Generally, filling pressure sets larger than holding pressure.

The crystalline resin like Iupital normally occurs large volume shrinkage during crystallization process. Holding pressure is necessary to fill the cavity with the resin and influences the mold shrinkage. Increasing holding pressure is effective for solving sink mark and void but excess holding pressure will be a cause of flash.

3.4 Injection velocity

It has better higher injection velocity for thin thickness molded part or multi cavity mold. On the other hand, it has better lower injection velocity for thick thickness molded part. Multi injection velocity control is effective for solving appearance problem e.g. jetting, flow mark.

3.5 Mold temperature

Mold temperature is one of important molding condition for Iupital injection. It is normally recommended 60～80℃. It is possible to set mold temperature 20~30 degrees-C for high-cycle operation molding.

However, it needs to pay attention to post shrinkage or residual stress.
High mold temperature over 120℃ is effective for improving surface appearance of molded parts. Mold temperature should to keep less than 150 degrees-C.

3.6 Molding cycle

3.6.1 Structure of molding cycle

Structure of molding cycle is as follows.

<table>
<tr><td colspan="2"></td><td colspan="7">Plasticizing</td><td></td></tr>
<tr><td colspan="2">Injection time</td><td rowspan="2">Cooling time</td><td colspan="7">Dry cycle time</td></tr>
<tr><td>Filling</td><td>Holding</td><td>Open</td><td>Eject</td><td>Take out</td><td>Insert</td><td>Interval</td><td>Close</td></tr>
<tr><td></td><td colspan="3">Necessary solidification time</td><td colspan="5"></td></tr>
</table>

Injection molding cycle ≧ injection time + cooling time + dry cycle time

3.6.2 Injection time (Filling time, Holding time)

Setting injection time is thought as below.

Injection time (Filling time + Holding time) ＞ Gate freezing time

Gate freezing time is the time that resin at gate portion will not able to flow by solidification. It is the time that when screw forward and holding time makes longer then the weight of molded part will be constant.

When injection time is shorter than gate freezing time then melted Iupital resin backflows through the gate. As a result, the dimensions and properties of molded parts are not constant to be maximum level and it will be cause of sink mark, void, or warpage.

Gate freezing time is easily determined by the examination of molded part weight vs. injection time. Several different injection time are set, 1-2 seconds apart, and molded parts weighed on a laboratory balance after removing sprue and runner. The weights of molded parts are plotted against injection time. The time that weight of molded parts will be constant is called gate-freezing time.

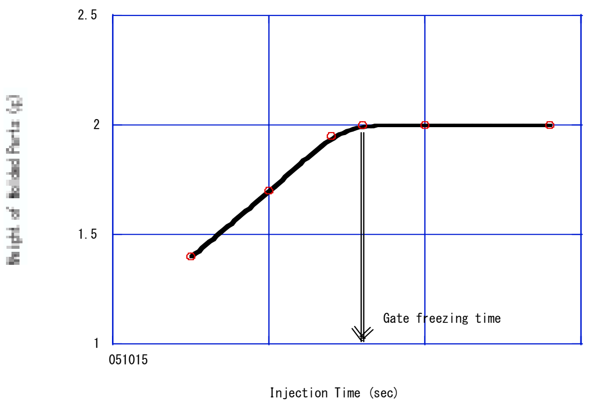


Figure-3.2 Gate freezing time

3.6.3 Plasticizing time

If plasticizing time is longer than minimum necessary cooling time, then molding cycle will be longer. In case of like this, it should better increase rotation number of screw or use injection machine having large plasticizing volume to reduce plasticizing time. Some special injection machine can operate plasticizing after mold open.

3.6.4 Cooling time

Necessary solidification time is a minimum cooling time that the molded part takes out from mold without any deformation after completion of holding time. Naturally cooling time is depending on wall thickness of molded parts, draft angle, type of ejection, location of ejection, holding pressure and mold temperature, etc.

Relation between wall thickness of Iupital molded part and necessary solidification time at each mold temperature calculated by computer simulation is shown in Figure-3.3. Necessary solidification time is able to estimate using this figure.

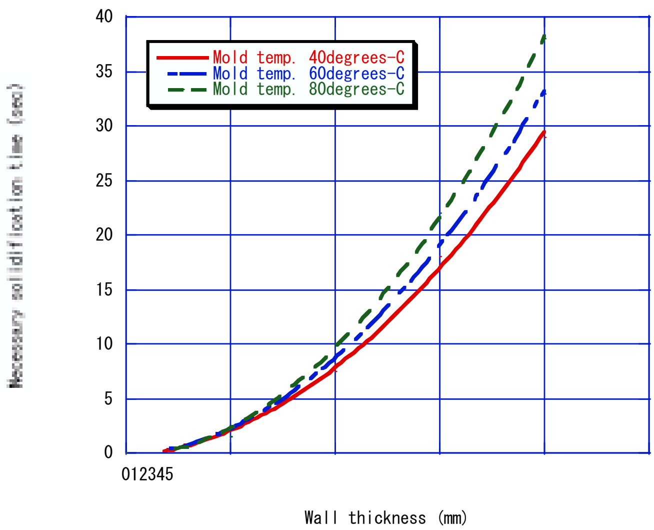


Figure-3.3 Relation between wall thickness and necessary solidification time

3.6.5 Dry cycle

Dry cycle is total mechanical time consisted of opening and closing mold, ejection, taking out time and interval time.

3.7 Resin replacement, temporary shut-down, and overhaul

3.7.1 Resin replacement

When replace the material in heating cylinder from Iupital to another material which has possibility of degradation at setting temperature of Iupital, has possibility to decompose Iupital and has much different molding condition from Iupital, it is desirable that Iupital is replaced by polyethylene or polystyrene which have wider molding temperature range. When replace from another material to Iupital, it is also desirable that to take same operation.

3.7.2 Temporary shutdown and overhaul

When work is interrupted, it is desirable to purge in the heating cylinder completely and to maintain setting temperature of the heating cylinder at 150℃ or lower due to safety reason or to prevent carbonized material contamination.

Incase of long term shutdown, it is necessary to replace to another resin like polyethylene or polystyrene.

During long time operation, decompose layer of resin is made at inside of cylinder, and gradually be carbonized. Molded part will be contaminated these carbonized layer of resin, during long time injection. Therefore, it is desirable to overhaul to prevent burnt occasionally.

3.8 Safety cautions

It must avoid to heat up Iupital over 230 degrees-C or to retain long time in injection cylinder at over 200 degrees-C. When Iupital retains in cylinder long time at high temperature, Iupital also will generate formaldehyde gas, as same as other polyacetal, because of thermal degradation. Therefore, it should better install standard exhaust system in the injection factory.

4. Quality of mold products

4.1 Dimensional accuracy (tolerance)

The table of SPI dimensional tolerance can be referred as standard tolerance of Iupital. Since dimensional accuracy is decreased, as the number of cavity increases, it should be considered.

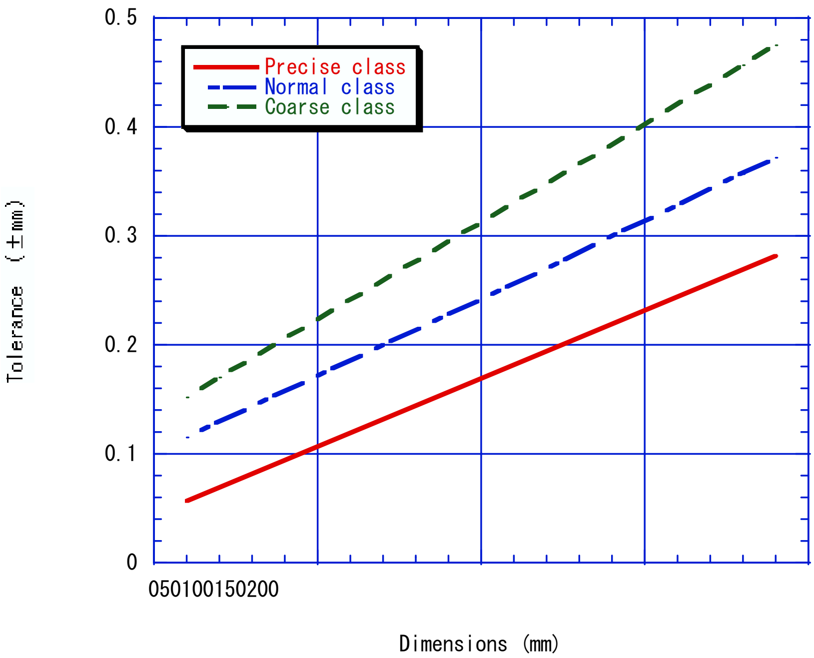


Figure-4.1 Dimensional tolerance (SPI)

4.2 Dimensional stability

In order to obtain the mold products with stable dimension, it is important for crystalline resin Iupital to keep constant temperature. This temperature means not only mold temperature and resin temperature but also oil temperature of injection machine and environmental temperature of mold factory.

Mold shrinkage of POM, depending on the mold temperature, is necessary to get stable measurement result over 48 hours.

When want to know the dimension is in tolerance or not, it is able to control simply to weigh molded part after injection, if relation between dimension and weight of molded part as experimental result is confirmed in advance.

4.3 Trouble shooting in Injection molding of Iupital

Trouble shootings in injection molding of Iupital are as follows.

Table-4.1-1 Trouble shooting in injection molding of Iupital

Trouble	Possible cause	Suggested Remedies
Silver streak	Moisture in the pellets. Air involved during plasticizing. Decomposition of resin by over-heating in cylinder, etc.	Preheat resin (80-90℃,3~4 hours). Increase backpressure. Decrease cylinder temperature. Clean up and start again.
Sink mark Voids	Insufficient filling of resin.	Increase injection-holding pressure. Increase pressure-holding time. Check the gate freezing time. Increase gate size. Check cushion stroke in injection. Check leaking of back-flow valve. Use high viscosity grade. Change gate location.
Flash	Insufficient clamping force of mold. High injection pressure/speed. Wear or deformation of mold. Low melt-viscosity of resin.	Increase clamping force of mold. Use large size injection machine. Decrease injection/holding pressure. Decrease injection speed. Repair or renew mold. Use high viscosity grade.
Record groove	Slow injection speed. Low injection/holding pressure. Low cylinder/mold temperature Insufficient flow length of resin and/or gate size.	Increase injection speed. Increase injection/holding pressure. Increase cylinder/mold temperature. Enlarge gate size.
Short shot	Low cylinder temperature. Low mold temperature. Low injection/holding pressure. Slow injection speed. Insufficient flow length of resin and/or gate size.	Increase cylinder temperature. Increase mold temperature. Increase injection/holding pressure. Increase injection speed. Use high flow grade.
Flow mark Jetting	Injection speed too high. Low mold temperature. Gates too small. Low cylinder temperature. Wrong gate location.	Decrease injection speed at gate. Enlarge gate size. Change gate location. Increase cylinder/mold temperature Use high flow grade.

Table-4.1-2 Trouble shooting in injection molding of Iupital

Trouble	Possible cause	Suggested Remedies
Warpage Deformation	Insufficient or excess injection/holding pressure. Non-uniform thickness of molded parts Non-uniform mold temperature. Non-uniform internal pressure. Shrinkage difference between flow direction and transverse direction.	Increase or decrease injection/holding pressure. Consider uniform thickness design. Use mold temperature controller, especially consider number and location of cooling channel. Change gate location, enlarge gate size
Weld mark Weak weld	Insufficient filling at weld parts. Disturbing welding at weld part. Insufficient gas ventilation at weld parts	Increase cylinder/mold temperature. Increase injection speed. Change gate location. Avoid mold release spray. Avoid resin contamination. Set/enlarge air vent. Change air vent location.
Sticking in mold Warping at mold release	Resistance to release because of high injection pressure/speed. Insufficient cooling time. Insufficient/incorrect releasing pin, device, etc. Undercuts and other unsuitable design.	Decrease injection pressure/speed. Increase cooling time (cycle time). Correct or change releasing way. Correct design of product. Use mold release spray temporarily.
Dimensional variation by shots	Insufficient resin melting. Shortage of injection /holding time. Unstable cushion stroke. Inconstant injection cycle. Unstable mold temperature.	Increase cylinder temperature. Extend injection/holding time. Maintain uniform cushion stroke. Maintain uniform molding cycle. Use mold temperature controller.
Yellow streak or black spot	Decomposition of resin by over-heating in cylinder. Long retention in cylinder. Air involved during metering.	Check the inside of cylinder and nozzle parts. Decrease cylinder temperature. Increase backpressure.
Brown or black marks	Insufficient air-vent in mold.	Set air-vent in mold. Decrease injection speed.
Unusual spot Smear	Contamination of other resins or foreign materials.	Clean up from hopper to nozzle. Change lot or bag of resin.

5. Thermal stability

5.1 Thermal analysis

As one of the evaluate method for thermal stability of resin usually use TGA (Thermo gravimetric Analysis) method, which analyze behavior of weight decrement of polymer degradation at melting state in high temperature. TGA curve (**Figure-5.1**) shows Iupital has excellent thermal stability.

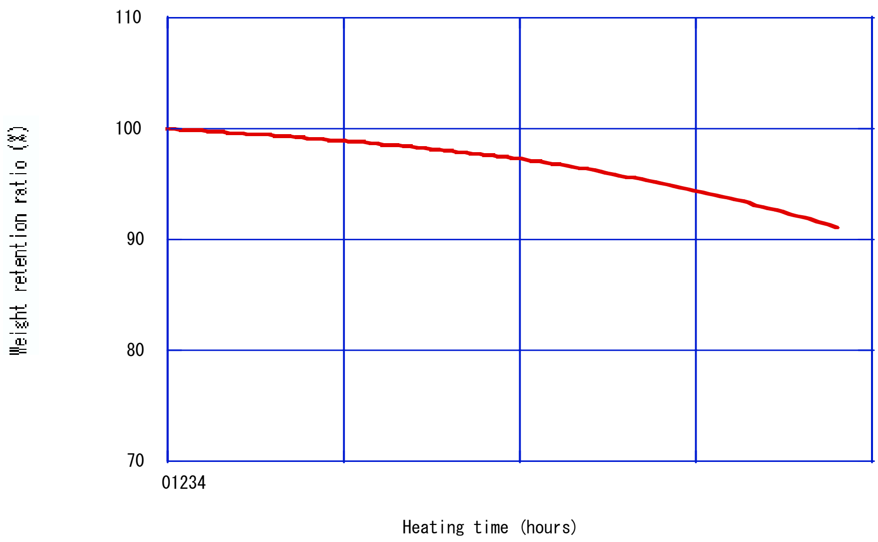


Figure-5.1 The characteristic of thermo gravimetric analysis (Standard grade)

5.2 Discoloration of retention in cylinder

Figure-5.2 shows discoloration characteristic curve of retention in cylinder. This figure means relation between resin temperature and retention time, when resin discolored △E=2.0 from the resin color of 190 degrees-C condition.

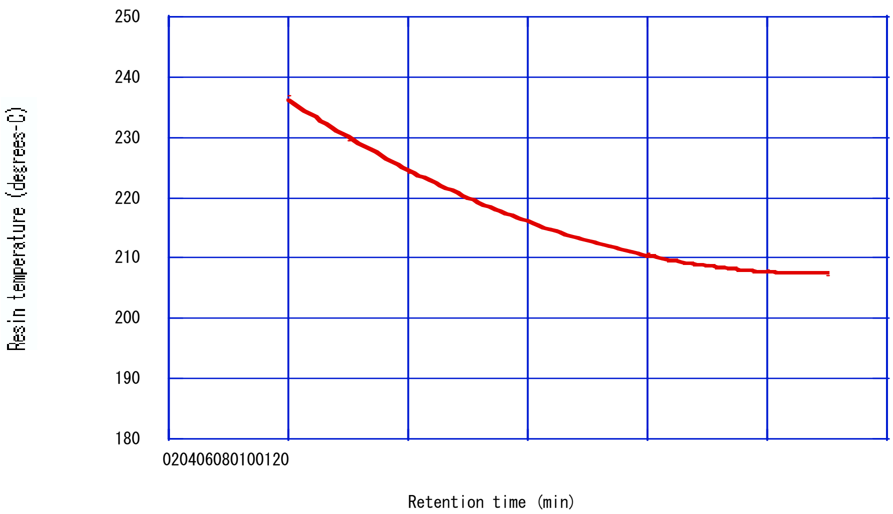


Figure-5.2 Discoloration curve of retention in cylinder

5.3 Recycle characteristics

Iupital can be recycled, since it has excellent thermal stability. **Figure-5.3.1, Figure-5.3.2, Figure-5.3.3, and Figure-5.3.4** show recycling ratio dependence of tensile strength, tensile elongation at break, mold shrinkage, and discoloration of Iupital standard grade respectively.

Physical properties of recycling Iupital 10 times repeated (recycle ratio 100%) is nearly equal to those of virgin Iupital, but the color and mold shrinkage slightly change. Less than 50% ratio of recycling has nearly same properties as virgin material.

However, recycling of reinforced grades by filler is not able to recommend because of the properties such as mechanical etc. decrease dramatically due to breakage of including filler. **Figure 5.3.5** shows recycle characteristics of POM glass filled grade.

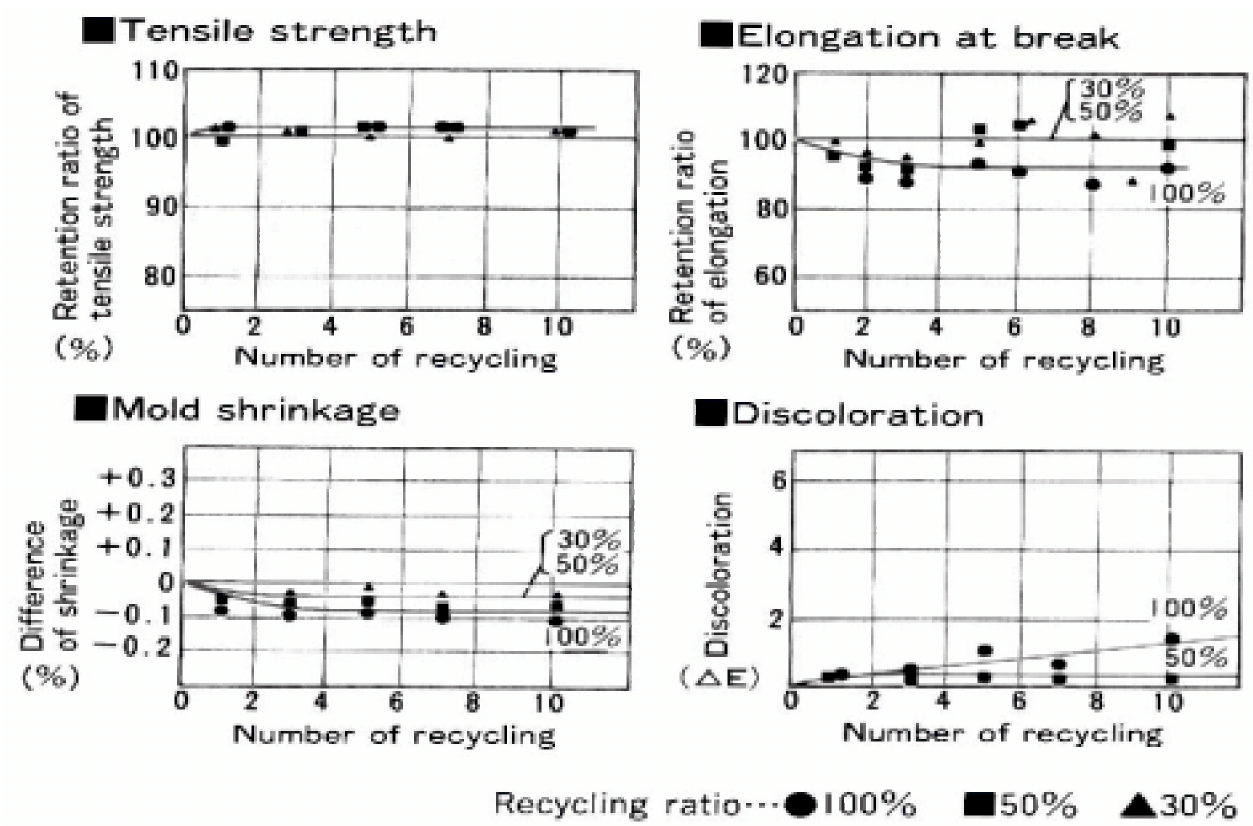


Figure- 5.3-1, -5.3-2, -5.3-3, - 5.3-4 Recycle characteristics of Iupital (standard grade)

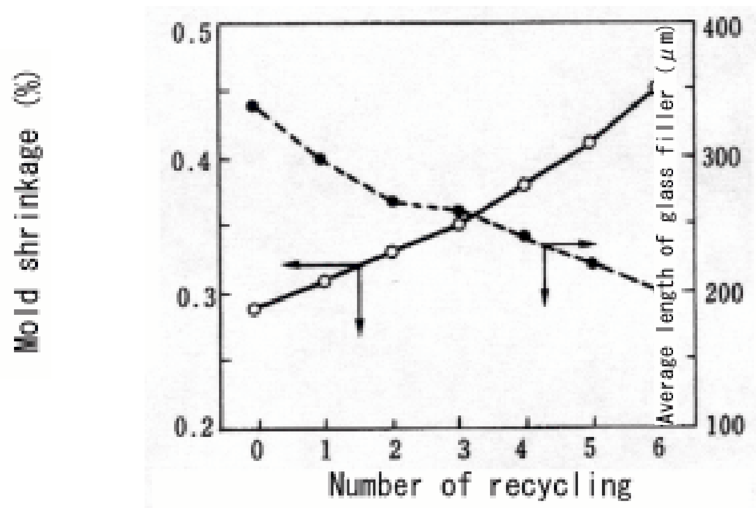


Figure-5.4 Recycle characteristics of POM glass filled grade

5.4 Mold deposit

Mold deposit (MD) means that adhered contaminant to mold surface during injection molding. In case of POM, if MD generate too much, MD causes non-good dimensional accuracy, insufficient mold release, and poor surface appearance.

5.4.1 Origin of MD

- Bleed out of additive substance of POM adheres to the mold surface.
- Formaldehyde (HCHO) gas generated from melted resin polymerizes to Paraformaldehyde [$HOCH_2O(CH_2O)nCH_2OH$] and adheres to the mold surface.

5.4.2 Relations between molding conditions and generation of MD

(1) Resin temperature

Generation of MD decreases with decreasing resin temperature.

Table-5.1 Influence of molding condition (F20-03): relative value as total of molding condition 1 is 100

	Solid of MD	Gas of MD	Total
Molding condition 1	10	90	100
Molding condition 2	4	36	40

	Cylinder temperature	Injection speed
Molding condition 1	230℃	85%
Molding condition 2	195℃	50%

(2) Injection speed

Too fast injection speed causes MD. (If injection speed is too fast, discharge air in cavity is not enough; insulated compression air in cavity generates high heat. High heat leads POM to heat degradation, and then generate MD.)

(3) Retention resin in cylinder

MD decreases with decreasing retention time. It is necessary to keep well balance shot capacity of injection machine and weight of products.

(4) Mold temperature

MD decreases with increasing mold temperature.

Table-5.2 Influence of mold temperature (estimated by gear type mold parts)

: F20-03 non pre-drying

	Shot number of MD generating
Mold temperature, 25℃	400
Mold temperature, 90℃	1400 (non-generated)

Cylinder temperature : 230℃ Injection speed : 85%

(5) Pre-drying condition of resin

Since moisture of resin helps the generation of MD, resin must be pre-dried sufficiently.

Table-5.3 Influence of pre-drying: F20-03 Relative value when result of dried is 100

Dry condition	Relative value of MD generating
Non-dried (water content: 0.4%)	118
Dried (water content: 0.0%)	100

5.4.3 Removal of MD and how to keep mold parts

(1) To clean up mold parts frequently

(2) Do not to keep mold parts with MD while storage (MD will cause rust of mold)

(3) If MD adhere to mold parts,

- remove MD by soft material against mold
- use metal polisher
- use removal spray for MD

(4) About removal spray for MD

When MD begins to generate, use the MD removal spray immediately. It will prevent to generate MD on the mold surface. However, after generate too much, it is not so effective to clean up the mold.

6. Product design

In order to fulfill the required performance of products, it is necessary to synthetically consider physical properties, mold ability, flowability, and the condition in designing mold. Basic criteria of mold design are as follows.

(1) Keep uniform wall thickness (**Figure-6.1**)

The non-uniform wall thickness of mold product and the extreme change of thickness disturb flow of resin, therefore appears flow mark, occurs warpage because of non-uniform shrinkage, or occurs deformation because of unbalanced cooling for resultant products.

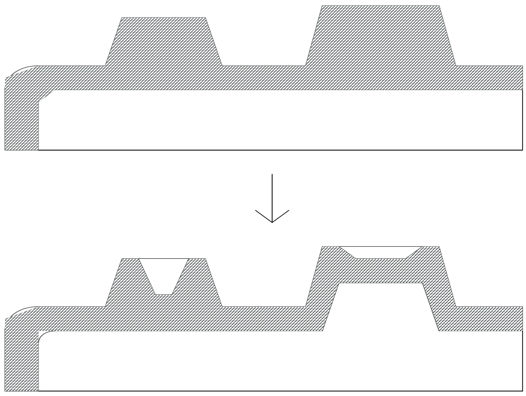


Figure-6.1

(2) Avoid undercut (**Figure-6.2**)

If mold product has undercut, it is necessary to arrange the slide core for mold release. Consequently, complex structure of mold makes it expensive.

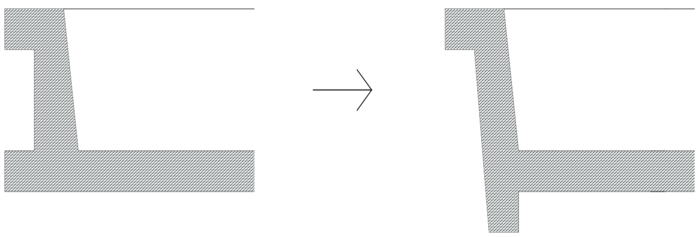


Figure-6.2

(3) Avoid sharp corner (**Figure-6.3**)

Since sharp corner disturbs flow of resin and to be a cause of flow mark. In addition, sharp corner tend to cause decrease of strength by generating notch effect or residual strain.

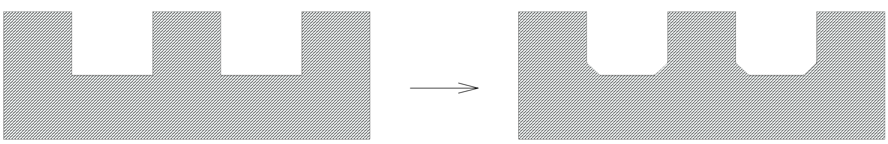


Figure-6.3

(4) Consider draft (**Figure-6.4**)

Iupital has good frictional properties even with mold. However, if draft is not sufficient, sometimes, mold products tend to be deformed by ejection. Therefore, it is better to give enough draft for mold design as possible.

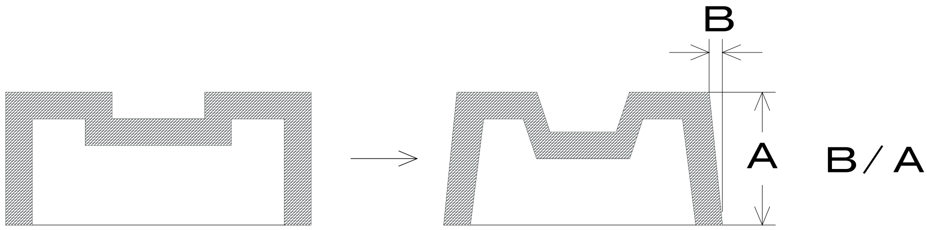


Figure-6.4

(5) Moderate wall thickness (**Figure-6.5**)

If the wall thickness of mold product is too thick, bad phenomena such as sink mark and void occur and long cooling time makes long molding cycle.

Figure-6.5

(6) Suitable thickness of rib (**Figure-6.6**)

When wall thickness cannot be thinned in the view of strength, uniform wall thickness is obtained by using rib.

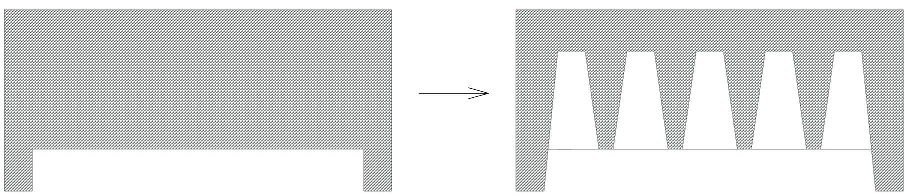


Figure-6.6

(7) Tough form of mold products (**Figure-6.7**)

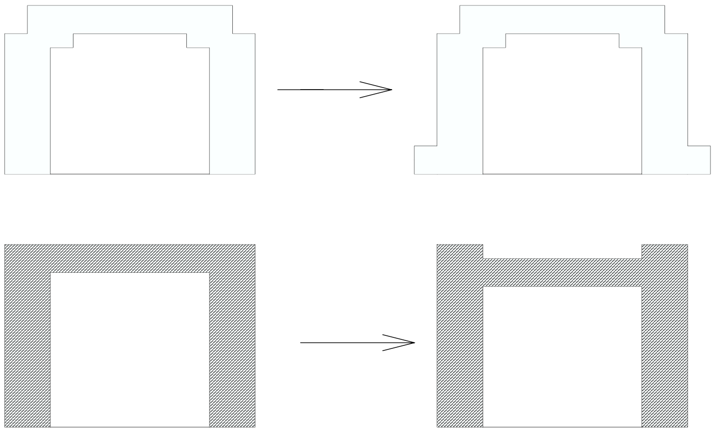


Figure-6.7

(8) The ratio of diameter to length of cylindrical mold products should be small.

The core pin with long and slender shape tends to be broken by the molding pressure. Since the temperature of the middle part of core pin becomes much higher than that of the edge, the variation of internal radius become wide. In case of high-cycle operation molding, internal skin core layer is broken by resin pressure.

(9) Consider making mold easily. Consider mold design for simple processing and easy finishing.

(10) Consider simple design for mold assembling and secondary processing.

(11) Consider position and direction of gate (**Figure-6.8**)

Gate position and number of gate are decided after considering weld position and resin orientation.

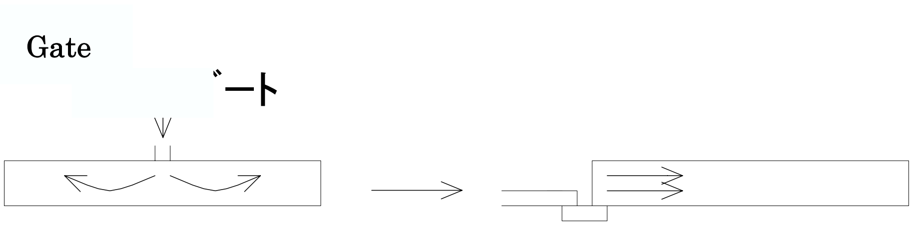


Figure-6.8

7. Mold design

The mold for injection molding must be designed to be able to show the required shape of mold products, number of mold products, and properties. It has important function that decides the productivity and quality of molded product, also.

7.1 Cavity

Since the heating and cooling of mold is quite important, it is necessary to separate the heating/cooling channel into some parts in order to obtain good temperature control and uniform temperature distribution. The cavity must be fully polished, because molded products are produced by replicate it. Ejection mechanism is enough as same as for other plastics.

7.2 Composition

In case of multi-cavity mold is used, dimensional variation will be large, the runner design for simultaneous filling is important. The mold having different cavity forms is not recommended. Cavity arrangement should be decided by considering the gravity. When the cavity consists of 2 products, the horizontal arrangement is better than the vertical one.

In case of mold product with thick wall thickness, the gravity causes bad phenomenon like jetting in the lower part of cavity. In case of a multi-cavity mold, the runner arrangement of **Figure-7.1 (B)** and **(C)**, having well-balanced runner length, is better than that of **Figure-7.1 (A)**.

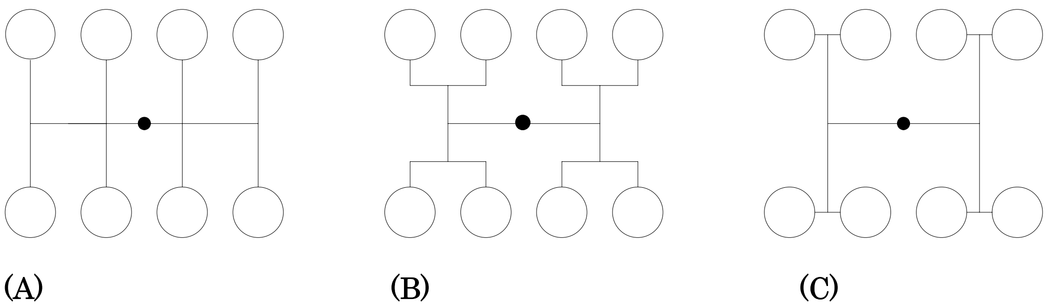


Figure-7.1 Runner balance

7.3 Composition of each part

7.3.1 Sprue and runner

Draft of sprue is used ordinary one but scratch or gap, such as originated undercut to release direction, should avoid. **Figure-7.2** shows the cross section shapes of runner and can be used each design. It is necessary for preventing the inflow of non-melted resin to prepare cold slug well at the basal part of sprue and the branched part of runner. Hot-runner should be used with the attention of resin retention. Especially, in case of small part, it needs attention retention resin in hot-runner.

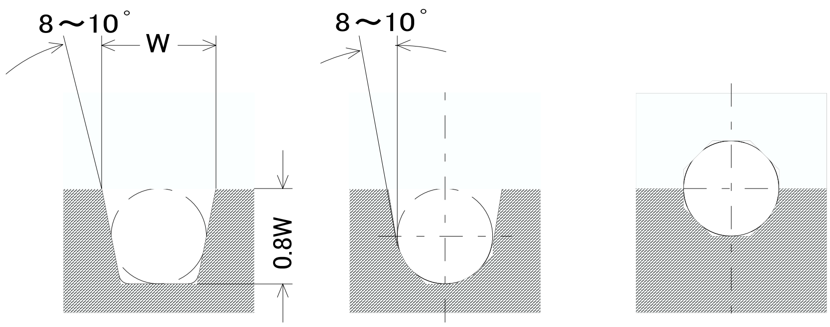


Figure-7.2 Cross section shape of runner

7.3.2 Gate

Gate design should be selected with considering several factors, e.g. number of products, performance, appearance, economic property, injection characteristic, etc.

(1) Direct sprue gate (**Figure-7.3**)

It is used in case of one cavity or put the gate at the bottom of molded part. It is easy to generate residual stress because injection pressure is applied to molded part directly, but mold structure will be simple.

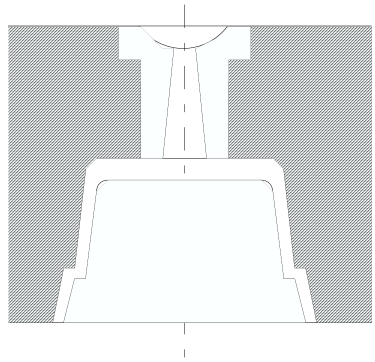


Figure-7.3 Direct sprue gate

(2) Side gate (**Figure-7.4**)

The side gate is used most typically. Normally, it is provided in the edge side of molded product and shape of cross section is rectangular or semicircular. It is suitable for multi cavity mold.

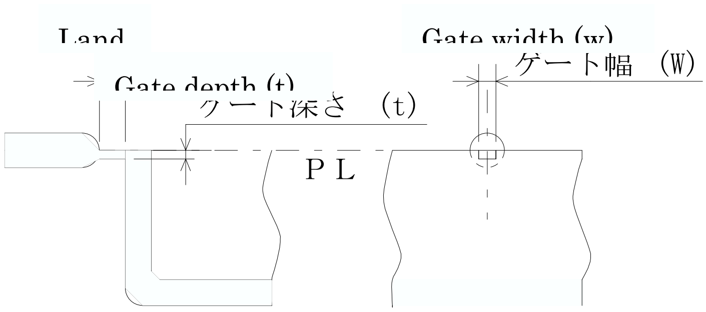


Figure-7.4 Side gate

(3) Fan gate (**Figure-7.5**)

It is similar to side gate but shape of the gate is like a fan. It is suitable for large size products.

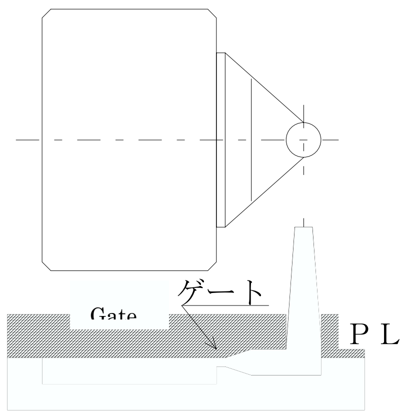


Figure-7.5 Fan gate

(4) Pin-point gate (Figure-7.6)

The diameter of pinpoint gate for POM is normally 0.5 to 2.0 mm. It is not necessary gate finishing. However, flow length will be decrease when the cross section of gate is small. It is easy to appear flow mark problem.

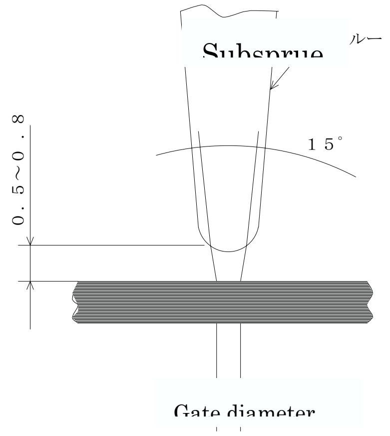


Figure-7.6 Pin-point gate

(5) Disk gate (Figure-7.7)

The disk gate is used for disk parts or cylindrical parts injection. This gate is effective to prevent the bad effect caused orientation of material flow. However, this gate has to cut and removed later in gate finishing process.

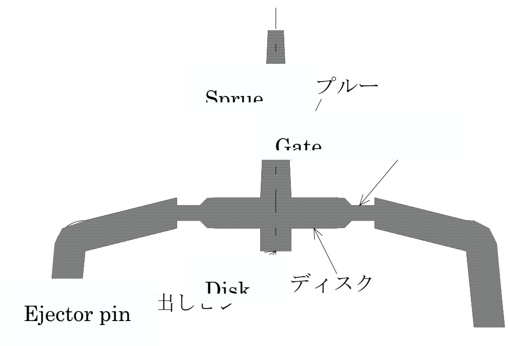


Figure-7.7 Disk gate

(6) Ring gate (Figure-7.8)

The ring gate is used for slender cylindrical products. It should consider ring gate shape that can fill resin into ring shaped runner first, because it will appear weld line.

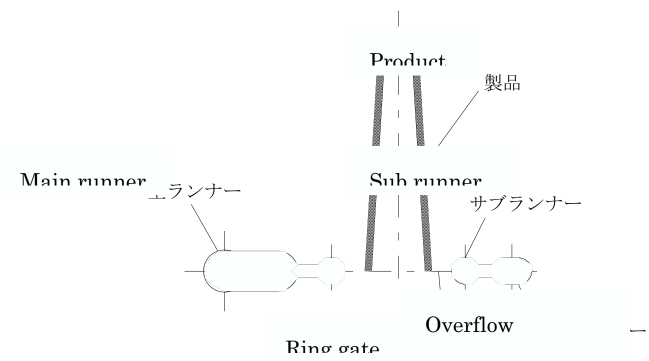


Figure-7.8 Ring gate

(7) Film gate (Figure-7.9)

The film gate is used for plate shaped products. It is also called flash gate or slit gate. Shape of film gate is a thin film with the full width of the molded part as shown in Figure-7.9.

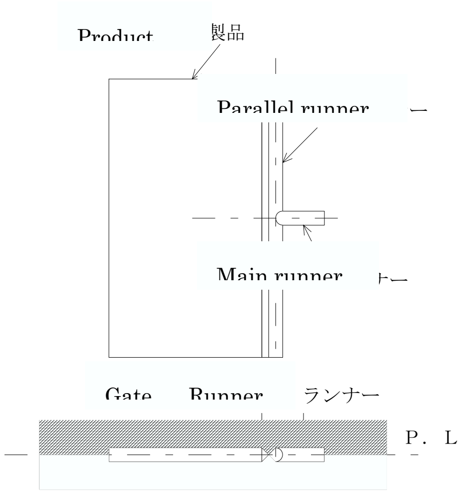


Figure-7.9 Film gate

(8) Tab gate (Figure-7.10)

The tab gate has a small additional tab in part of the molded product to which the melted material is conducted from another gate (primary gate) so that the material from the runner may not flow directly into the cavity.

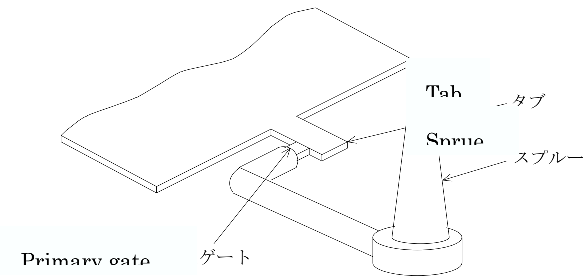


Figure-7.10 Tab gate

(9) Submarine gate (Figure-7.11)

The runner portion of the submarine gate is located on parting surface and the gate portion is connected at the side of molded product through a conical hole provided in the mold plate of stationary side or movable side.

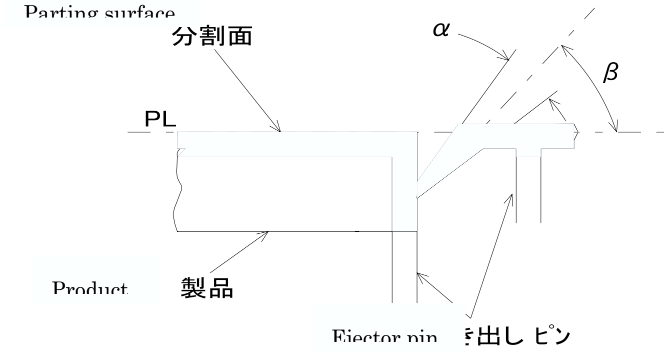


Figure-7.11 Submarine gate

7.3.3 Varying wall thickness (**Figure-7.12**)

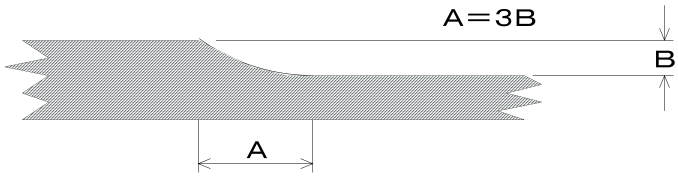


Figure-7.12 Varying wall thickness

7.3.4 Draft (**Figure-7.13**)

The draft should be to ensure easy ejection as **Figure-7.13**. Normally the draft angle is 0.5 ~ 1.0 degree enough.

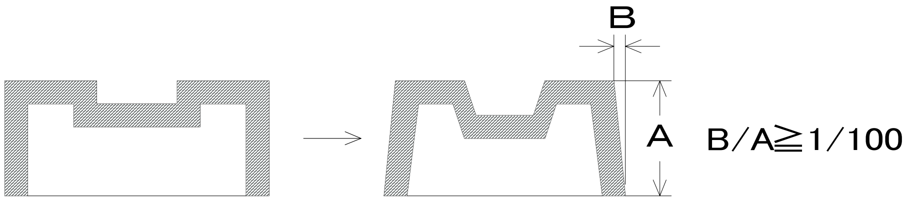


Figure-7.13 Draft angle

It is necessary more large draft angle for the grating portion. (**Figure-7.14**)

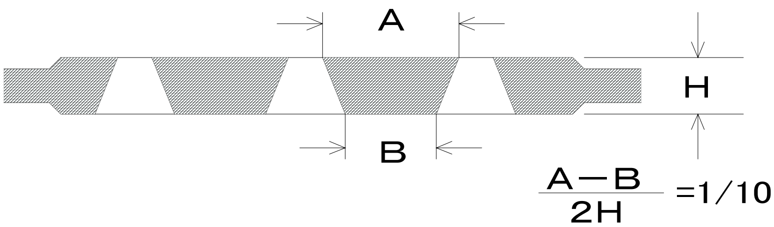


Figure-7.14 Grating portion

7.3.5 Undercuts (**Figure-7.15**)

As a rule, undercuts of molded products are not recommended. When undercuts cannot be avoided, it will be design as follows.

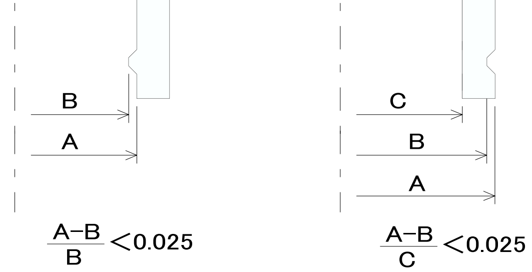


Figure-7.15 Undercuts

7.3.6 Corner radius: corner R (**Figure-7.16**)

If possible, it is to be desired that the corner radius (corner R) should consider at least larger than 1 mm R.

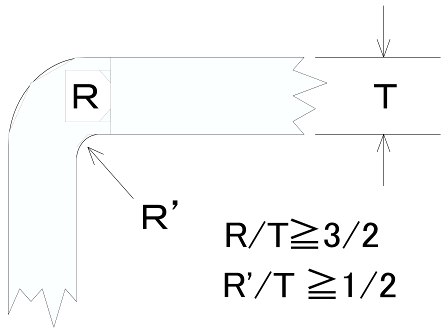


Figure-7.16 Corner radius (corner R)

7.3.7 Rib (**Figure-7.17**)

The proper rib proportion is shown **Figure-7.17**.

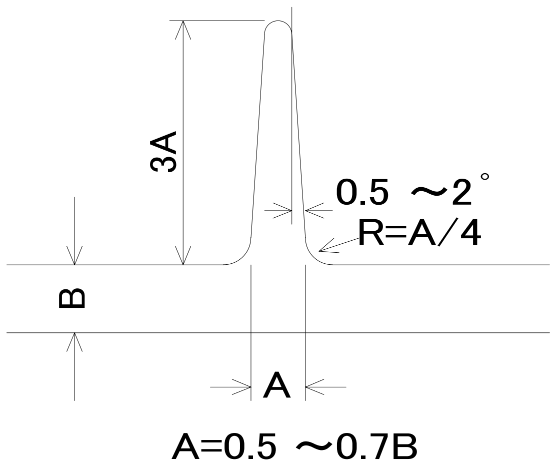


Figure-7.17 Typical design of rib for POM]

7.3.8 Boss (**Figure-7.18**)

The proper boss design is shown **Figure-7.18**.

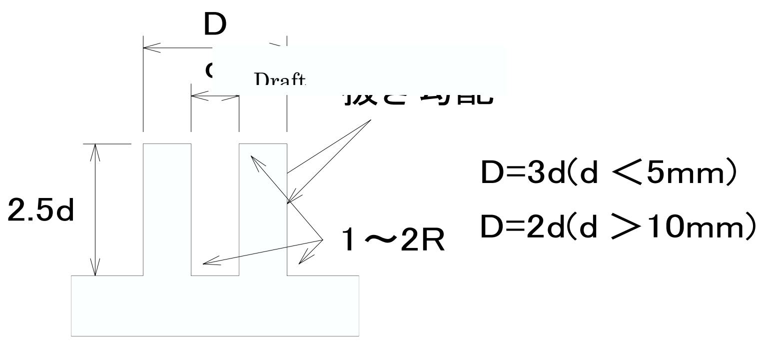


Figure-7.18 The proper boss design

7.3.9 Hole (**Figure-7.19**)

The design of hole is shown **Figure-7.19**. It is necessary to pay attention, when the hole located near at edge of product or another hole, there is some possibility that the strength of product will be weak.

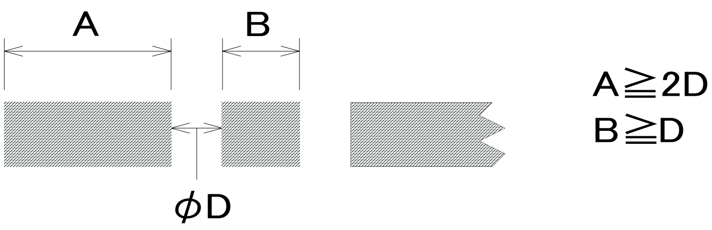


Figure-7.19 Design of hole

7.3.10 Air vent

The provided ways of air vent can be divided roughly into three.

1. Air vent located on the parting surface of the mold
2. Air vent located in cavity of the mold or core of the mold
3. Air vent by special method

(1) Air vent located on the parting surface of the mold

The depth of air vent is usually used 0.005~0.02 mm for POM resin. The location of air vent is provided such as;

1. As far from gate as possible
2. Weld line tends to appear
3. The end of runner or sub runner

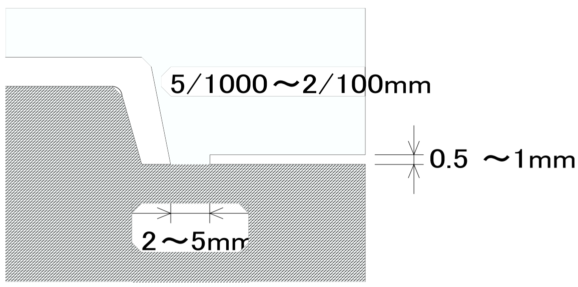


Figure-7.20 Air vent

(2) Air vent located in cavity of the mold or core of the mold

1. The method of using ejector pin (**Figure-7.21**)

This method uses clearance between ejector pin and hole of ejector pin. The clearance between ejector pin and hole of ejector pin is 0.02~0.03 mm for 5~10 mm diameter pin and 0.01~0.02 mm for less than 5 mm diameter pin typically.

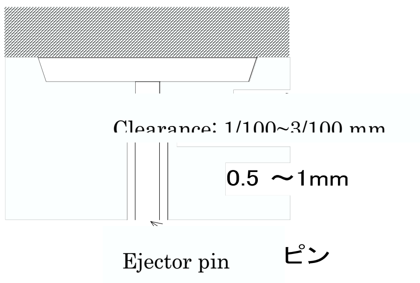


Figure-7.21

2. The method of using core pin (Figure-7.22)

It should better to provide air vent at around core pin, when the molded part has high bosses or ribs.

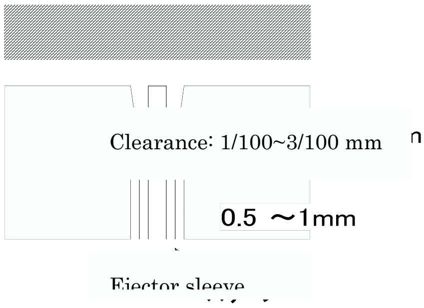


Figure-7.22

3. Using cavity or core plate

This method is used for venting of high rib. This method makes venting use of the clearance between thin stratified plates and others. Also can provide these thin stratified plate at cavity portion, then the air inside cavity is able to exhaust from cavity.

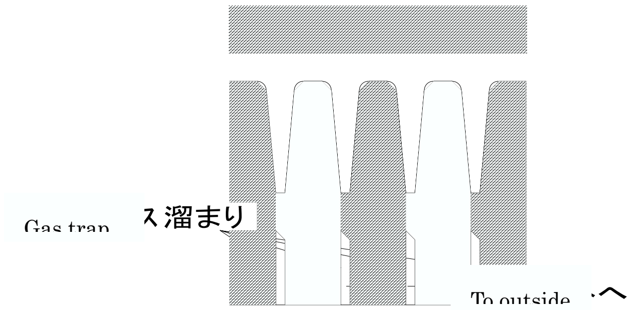


Figure-7.23

(3) Air vent by special method

As a special method of air venting, there is the way using vacuum pump for exhausting air of inside cavity. This method is almost ideal method for air venting, however the price of equipment will be expensive and the structure of mold will be complicated are disadvantages.

7.3.11 Tilt of core pin

Since slender core pin rarely happens titling by resin pressure, the ratio of diameter to length should better to be the ratio 1: 5 for the free edge core pin, and to be the ratio 1: 10 for fixed edge core pin.

7.3.12 Mold temperature control

The design of cooling channel for mold temperature control is very important because of the mold is a heat exchanger during injection molding. Normally, the cooling channel diameter of mold for Iupital is desirable 8~12 mm diameter. The cooling channel is desirable to locate as near cavity surface as possible, the distance between each cooling channel is located as closer as possible.

The minimum distance from cavity surface to cooling channel is depending on mold material, dimension of cooling channel, type of cooling channel, and internal pressure of cavity.

The relation between distance from cavity surface and dimension of cooling channel is shown **Figure-7.24**. This graph shows the calculation result, when inside pressure of cavity added cavity surface then cavity surface deforms 2 μ m. The distance of each cooling channel is 0.7 times of cooling channel diameter by empirical knowledge.

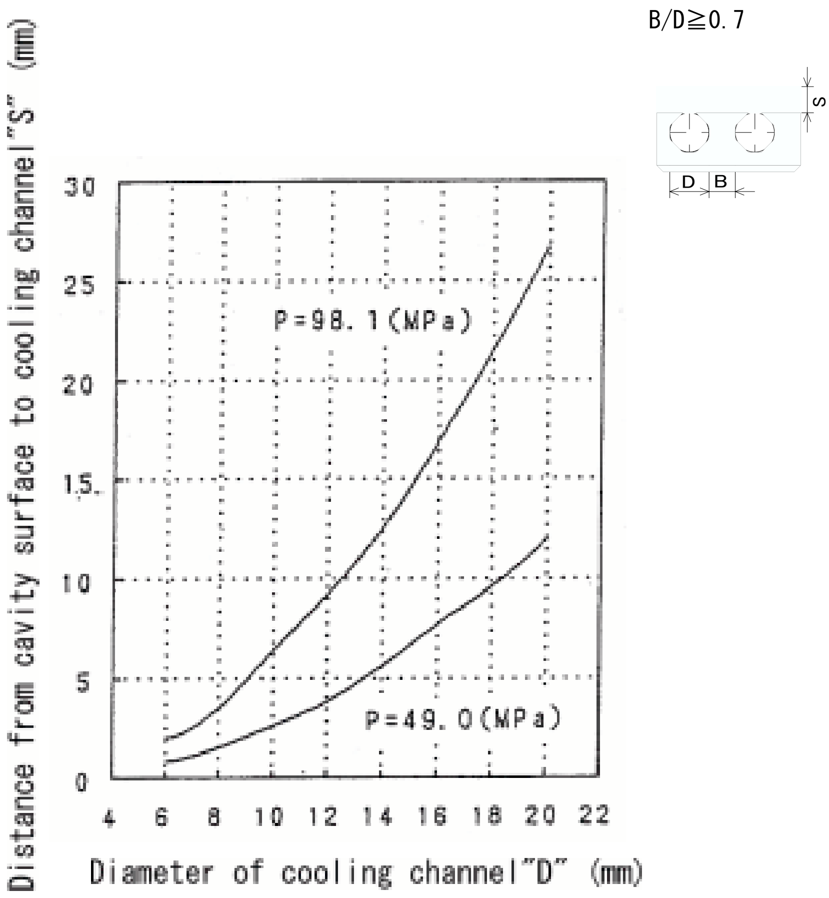


Figure-7.24 Relation between distance from cavity surface and dimension of cooling channel

However, it is difficult to keep such like distance at the actual mold. There are several combinations of cooling channel in actual mold. The kind of cooling channels are shown as follows. (Figure-7.25, Figure-7.26, Figure-7.27, Figure-7.28, Figure-7.29, Figure-7.30)

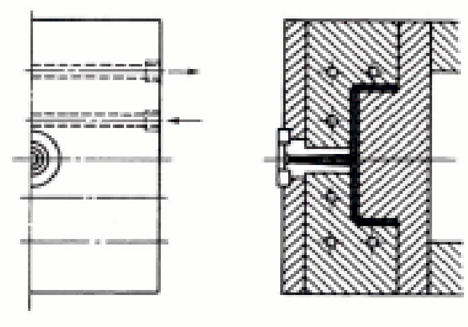


Figure-7.25

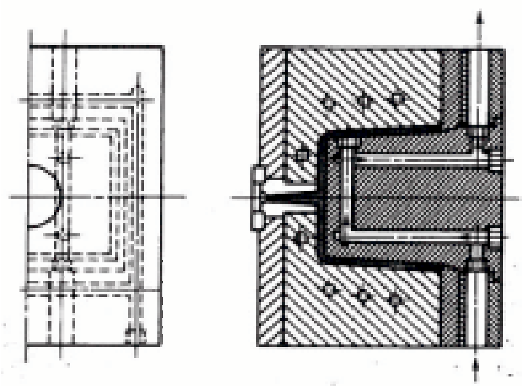


Figure-7.26

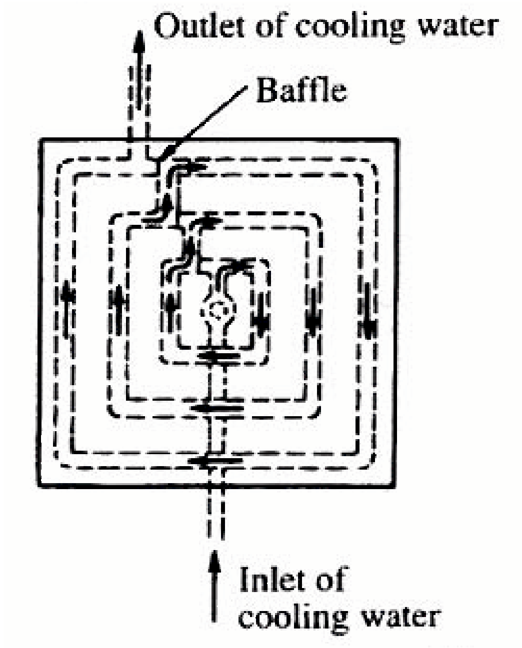
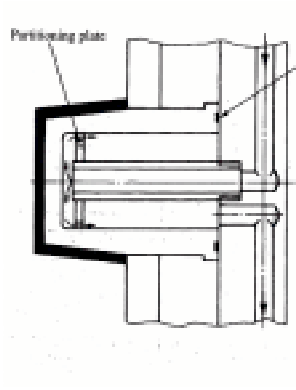


Figure-7.27



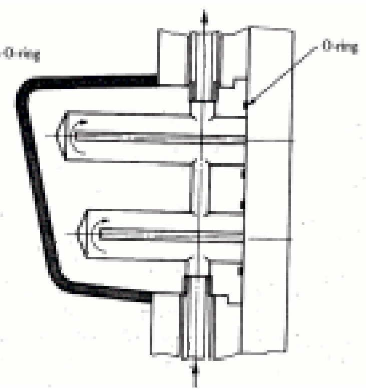


Figure-7.28 Figure-7.29

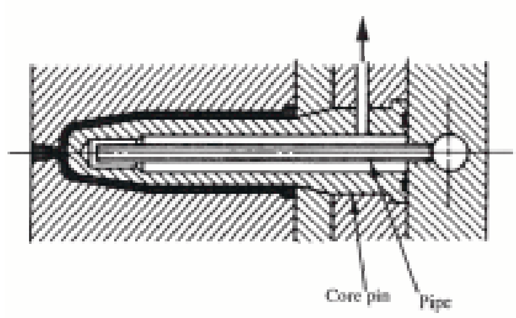


Figure-7.30

If it is impossible to provide cooling channel, it is possible to use well heat conductivity material for mold parts. As reference, thermal conductivities of mold part materials are shown Table-7.1.

Table-7.1 Thermal conductivity of mold part materials

Mold material	Heat conductivity	
	W/m・K	kcal/m・hr・℃
Steel (S50C 0.5%C)	53	46
SKD 61	34	29
Stainless steel (SUS304)	16	14
Zinc alloy (ZAS 4%Al, 3%Cu)	109	94
Cupper alloy (HR750: Kobe Seiko)	129	111
Belyllium copper alloy 20C	121	104
Belyllium copper alloy 275C	109	94

7.3.13 Mold parts materials

The required properties for injection molding molds are as follows.

1. Toughness and hardness
2. Good machining property
3. Excellent crepe finish quality and good finish surface after EDM.
4. Excellent mirror polish. Uniform structure, high surface hardness
5. Excellent corrosion resistance and rustproof
6. Good wear resistance
7. Good hardening properties and low heat-treatment deformation

The characteristics of mold parts materials are shown in Table-7.2. It is desirable to select mold parts material to fix each mold part required by referring these characteristics.

Table-7.2 Characteristic of mold parts materials

Type of meterial	Hardness (HRC)	JIS type	Machining quality	Mirror Polishining properties	Crepe-machining properties	Corrosion resistance	Wear resistance
Pre-hardened steels	14	S55C	10	4	6	2	2
	27	SCM445	8	6	7	3	3
	33	SCM improve	9	5	6	4	4
		SNCM improvee	8	9	9	4	4
	40	Precipitation-hardned	8	8	7	5	5
		Precipitation-hardned	7	9	10	5	5
	35,40	SUS630 improve	3	8	8	10	4
Hardning steel	53	SUS420J2 improve	7	10	9	9	7
	42～52	SKD61	8	8	7	6	7
	55～60	SKD11 improve	6	9	8	6	9
		SKD11	5	7	5	6	9
		SUS420 improve	7	7	7	5	8
		SUS440C improve	5	9	8	8	9
	63～68	Powder matal HSS	5	8	7	6	10
Maraging steel	55	Maraging steel	4	10	10	6	7
	45	Non-Magnetic steel	2	8	-	6	7

8 . Flow characteristic

8.1 Flowability of Iupital and other resins

The bar flow length of temperature dependence about Iupital and other kind of resins are shown in **Figure-8.1**.

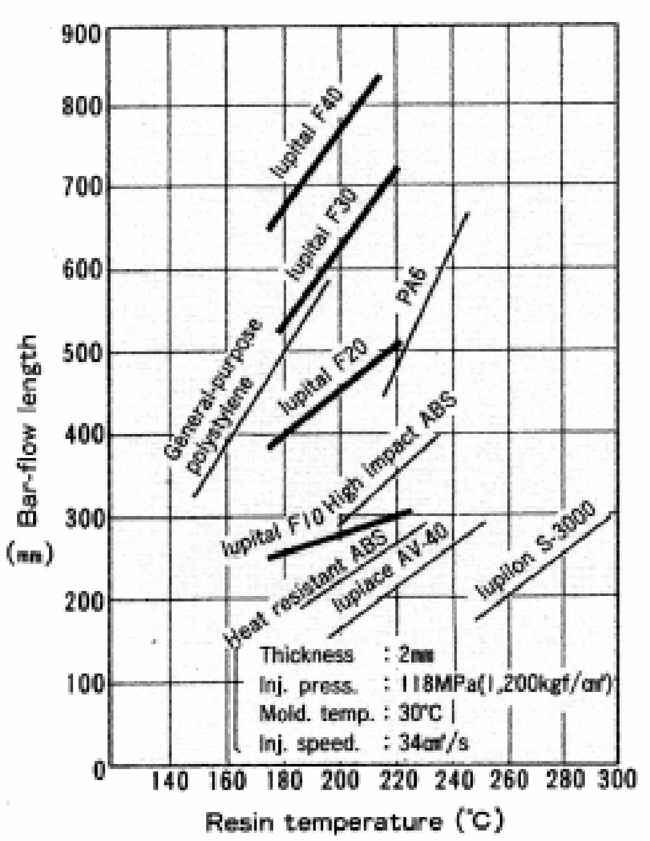


Figure-8.1 Bar flow length of Iupital and other kind of resins

8.2 Bar flow length depending on wall thickness

The relation between bar flow length and wall thickness is shown **Figure-8.2**.

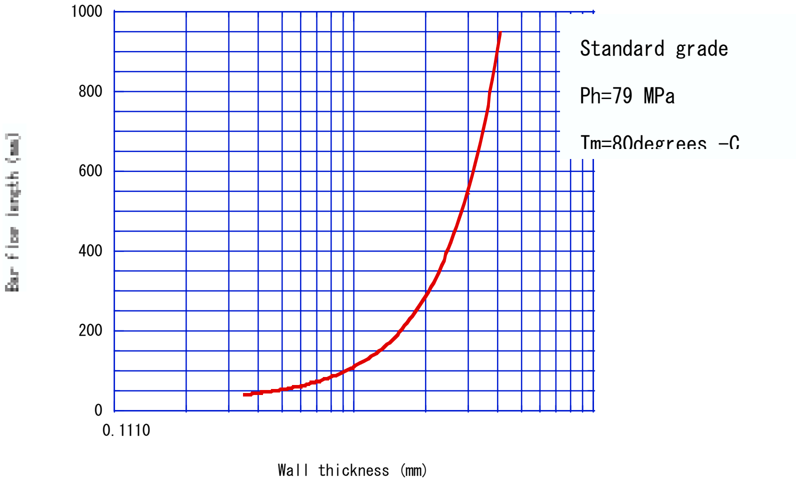


Figure-8.2 The relation between bar flow length and wall thickness

8.3 Bar flow length depending on molding conditions

Bar flow length shows various tendency depending on molding conditions. Relations between bar flow length and molding conditions as flow length ratio are shown as follows. Basic molding conditions are as follows.

Grade:	F20-03		
Holding pressure:	79 MPa	Mold temperature:	80 degrees-C
Resin temperature:	200 degrees-C	Injection velocity:	68 cc/sec

Bar flow length of basic condition is as L_0, then calculated flow length ratio L/L_0 for each molding conditions.

8.3.1 Holding pressure

Relation between holding pressure and flow length ratio is shown **Figure-8.3**.

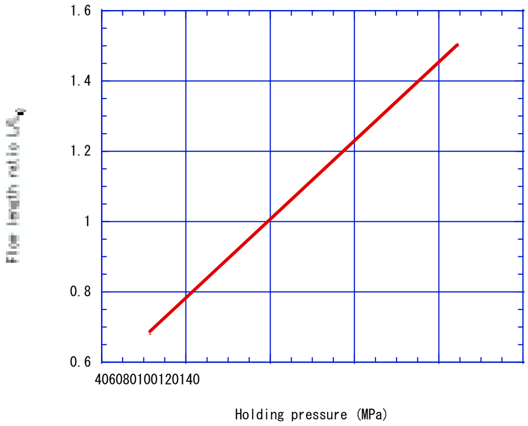


Figure-8.3 Relation between holding pressure and flow length ratio

8.3.2 Mold temperature

Relation between mold temperature and flow length ratio is shown **Figure-8.4**.

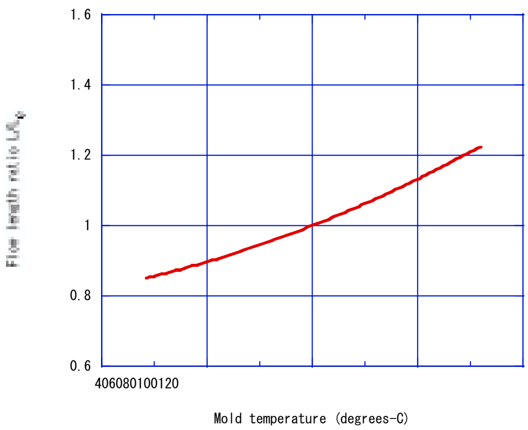


Figure-8.4 Relation between mold temperature and flow length ratio

8.3.3 Resin temperature (Cylinder temperature)

Relation between resin temperature and flow length ratio is shown **Figure-8.5**.

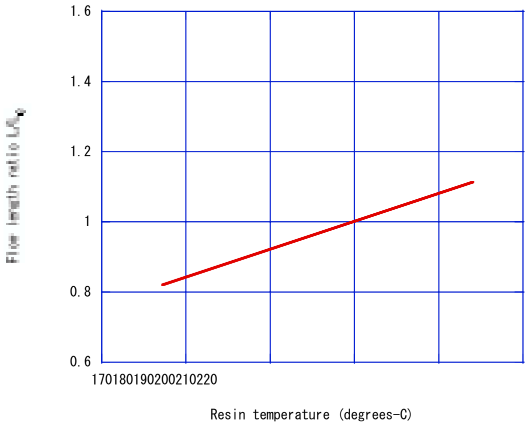


Figure-8.5 Relation between resin temperature and flow length ratio

8.3.4 Relation between Melt Flow Rate (MI) and bar flow length

Relation between Melt Flow Rate (MI) and bar flow length is shown **Figure-8.6**.

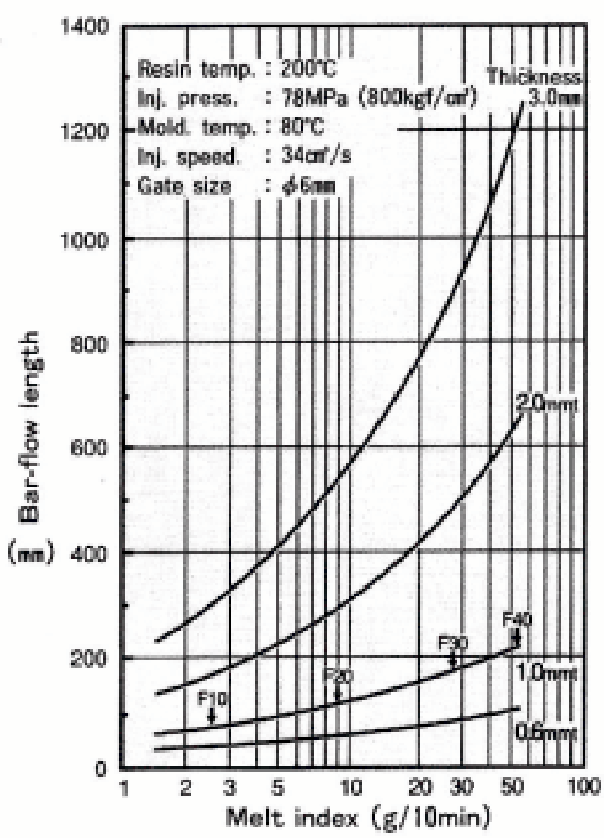


Figure-8.6 Relation between Melt Flow Rate (MI) and bar flow length

9. Characteristic of mold shrinkage

Mold shrinkage can vary with several factors. The factors are wall thickness, polymer melt viscosity, kind of grade, molding conditions such as mold temperature, injection pressure (holding pressure), injection time (holding time), injection velocity and cylinder temperature. Other factors such as mold design, gate size, flow length and flow direction; filler type and level also effect mold shrinkage.

9.1 Relation between thickness of products and mold shrinkage

Mold shrinkage changes depend on wall thickness of molded parts. 2 ~3 mm thickness indicate minimum mold shrinkage. The relation between wall thickness and mold shrinkage of Iupital standard grade is shown **Figure-9.1**.

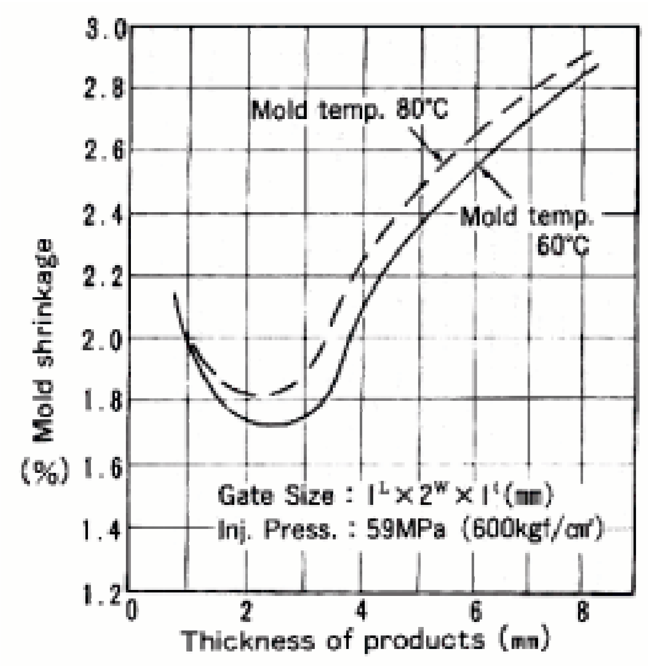


Figure-9.1 Relation between wall thickness of products and mold shrinkage

9.2 Relation between Melt Flow Rate (MI) and mold shrinkage

The relation between Melt Flow Rate (MI) and mold shrinkage is shown **Figure-9.2**.

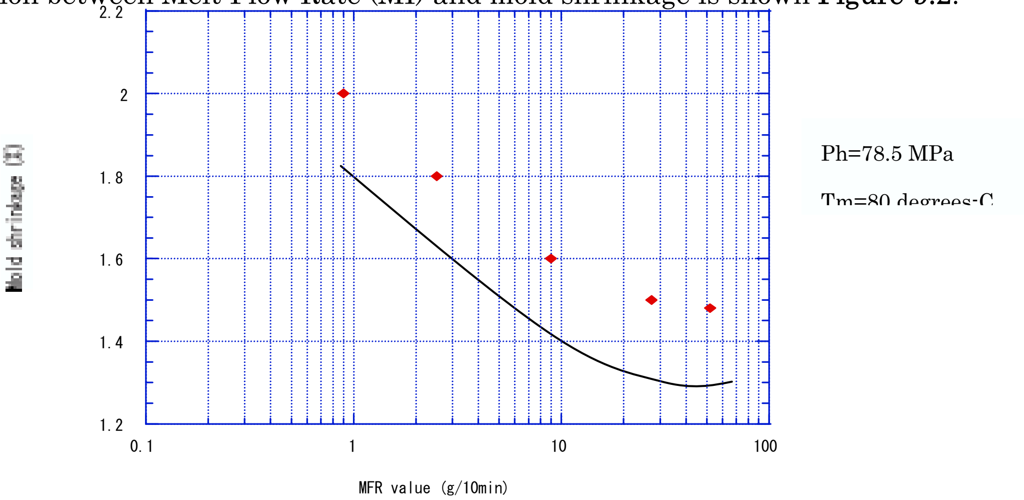


Figure-9.2 Relation between Melt Flow Rate and mold shrinkage

9.3 Difference of mold shrinkage

There are some difference of mold shrinkage between flow direction and transverse direction. It is necessary to keep attention about the difference of mold shrinkage because it sometimes shows much difference depend on the kind of grade. The difference of mold shrinkage is shown in **Table-9.1**.

Table-9.1 Difference of mold shrinkage

Grade	Flow Direction //	Transverse direction ⊥	Difference of shrinkage //−⊥
F10	1.8	1.8	0.0
F20	1.6	1.6	0.0
F30	1.5	1.5	0.0
F40	1.5	1.5	0.0
FV-30	1.5	1.5	0.0
FG1025A	0.6	0.9	-0.3
FG2025	0.5	0.8	-0.3
MF3020	1.5	1.0	0.5
FB2025	1.3	1.3	0.0
FC2020D	0.2	0.6	-0.4
FC2020H	0.3	0.6	-0.3
FT2010	1.4	1.4	0.0
FT2020	0.8	0.8	0.0
LO-21	1.6	1.6	0.0
FX-11	1.6	1.6	0.0
FL2010	1.6	1.6	0.0
FL2020	1.6	1.6	0.0
FW-21	1.6	1.6	0.0
FW-24	1.6	1.6	0.0
FM2020	1.6	1.6	0.0
FU2025	1.3	1.3	0.0
FU2050	0.8	0.8	0.0
ET-20	1.1	1.1	0.0
TC3015	1.2	1.2	0.0
TC3030	0.9	0.9	0.0

Holding pressure: 78.5 MPa
Mold temperature: 80 degrees-C
Cylinder temperature: 200 degrees-C
Test specimen: 4in, 3mmt, disk

9.4 Relation between molding conditions and mold shrinkage

(1) Holding (Injection) pressure

The relation between holding pressure and mold shrinkage of Iupital tends as shown **Figure-9.3**.

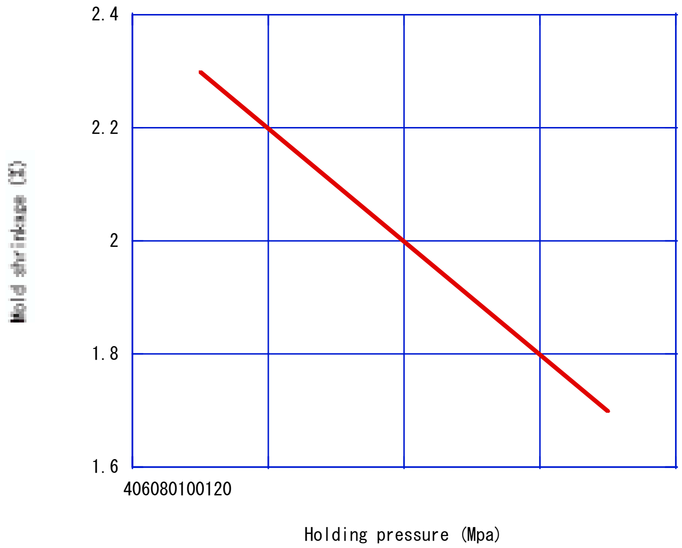


Figure-9.3 The relation between holding pressure and mold shrinkage

(2) Injection time

The relation between injection time and mold shrinkage of Iupital tends as shown **Figure-9.4**. After gate freezing, mold shrinkage shows constant value.

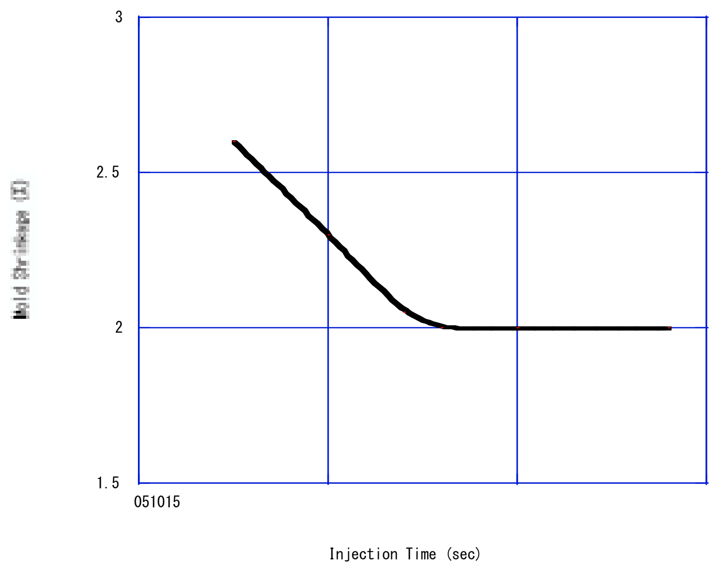


Figure-9.4 The relation between injection time and mold shrinkage

(3) Mold temperature

The relation between mold temperature and mold shrinkage tends as shown **Figure-9.5**. It tends nearly direct proportion.

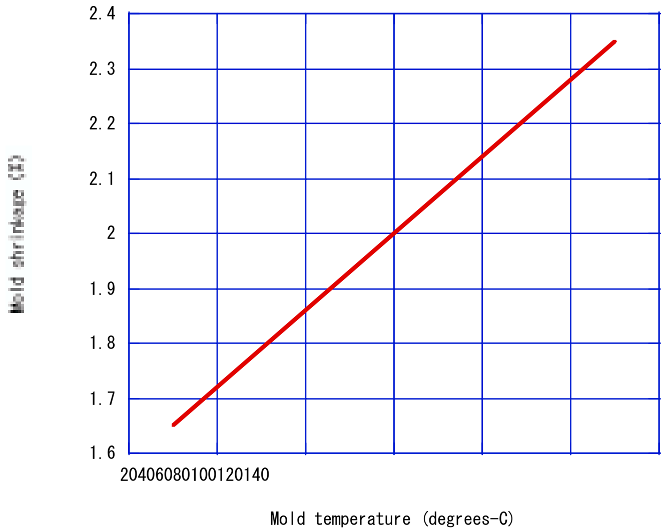


Figure-9.5 The relation between mold temperature and mold shrinkage

(4) Resin temperature (cylinder temperature)

The relation between resin temperature and mold shrinkage tends as shown **Figure-9.5**. It is not so effective comparing mold temperature.

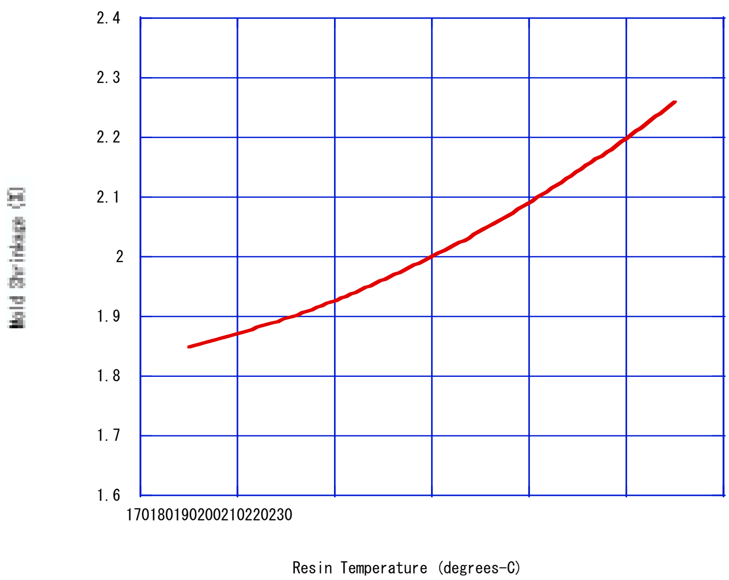


Figure-9.5 The relation between resin temperature and mold shrinkage

10. Characteristic of shrinkage due to heat aging

Iupital as crystalline resin occurs shrinkage due to heat aging because of re-crystallization by annealing or heating from the actual using environment. The dimension of molded parts changes depend on wall thickness, molding conditions, heating temperature and heating time.

The relations between shrinkage due to heat aging and heating time, wall thickness, heating temperature, are shown Figure-10.1, Figure-10.2, and Figure-10.3 respectively.

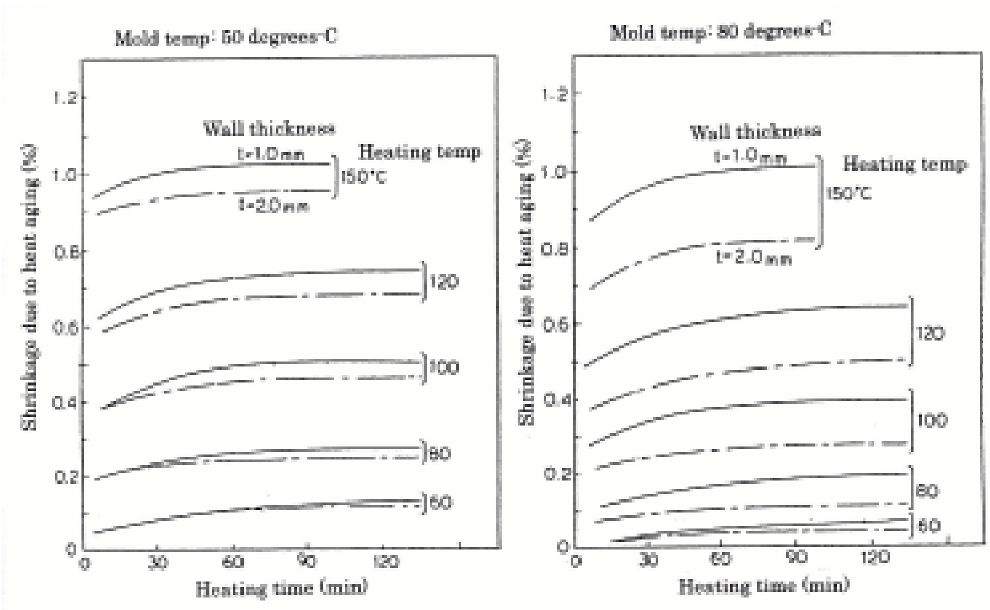


Figure-10.1 The relation between heating time and shrinkage due to heat aging

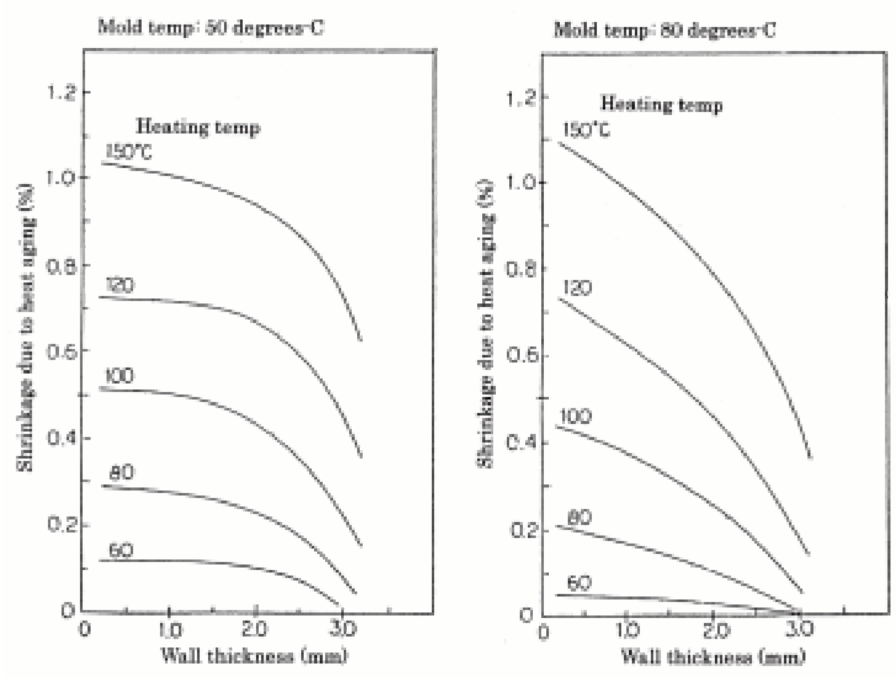


Figure-10.2 The relation between wall thickness and shrinkage due to heat aging

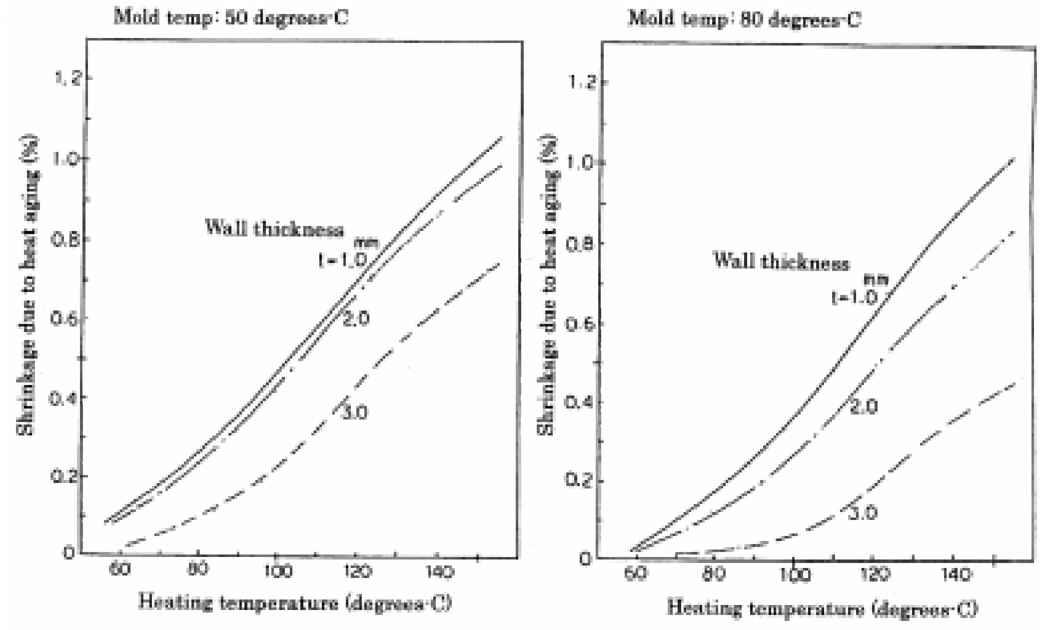


Figure-10.3 The relation between heating temperature and shrinkage due to heat aging

11. Post-molding shrinkage

The molded parts of POM occur dimensional shrinkage after injection because of crystallization at room temperature. Especially the molded part injected by lower mold temperature shows larger and longer time post-shrinkage. Characteristic of post-mold shrinkage for Iupital standard grade is shown Figure-11.1.

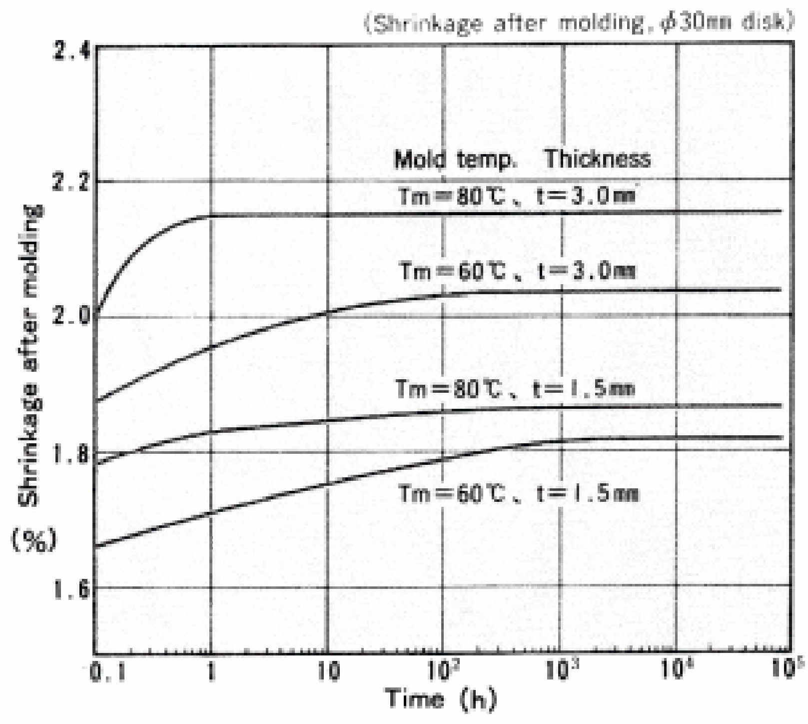


Figure-11.1 Post-mold shrinkage characteristic of Iupital standard grade

(References)

Polyacetal handbook:	Nikkan Kogyo Shinbun-sha
The bases for mold design:	Plastic Age
Injection molding:	Plastic Age
Mold for injection molding:	Nikkan Kogyo Shinbun-sha
Guide for mold designing:	Σ shuppan
Plastic mold handbook:	Japan synthetic plastic engineering association