1. Molding machine

Inline screw type is most common Lupital molding machine.

1.1 Molding machine selection by injection volume

Molding machine injection volume $Q(\text{gr})$ 1 shot weight $W(\text{gr})$, including the sprue and the runner, within a following range is preferable.

$$Q = (1.3 \sim 1.5) \times W$$

If the injection volume is too small, plasticization will not make it, and Lupital will be sent to the end of the screw before sufficient plasticization, which will end up with losing physicality it would had as molded product. Conversely, if the injection volume is too big, retention time in cylinder will be long, and will likely to cause degradation of resin.

1.2 Molding machine selection by mold clamping pressure

When molding Lupital, both toggle type and direct pressure type is fine.

Relationship between molded product projected area $A(\text{cm}^2)$ and required clamp capacity $P(\text{ton})$ shown below is preferable.

$$P = (0.5 \sim 0.7) \times A$$

1.3 Nozzle structure

Open nozzle is commonly used for Lupital molding.

There are (A) open nozzle, and (B) shut-off nozzle as in Figure-1.3-1, for the nozzle of commercially available injection machine, but with either nozzle, it must have temperature control function.

![A: Open nozzle, B: Shut-off nozzle](image)

Figure-1.3-1  Molding machine nozzles

If concerning about drooling from nozzle, use shut-off nozzle. However, it might cause silver streak or burn from Lupital retention in slide part, so extra attention is required.

1.4 Injection mechanism

Normal molding machine, "determinate injection speed, and injection pressure dual-regulation", is sufficient for molding Lupital, but when molding a product which requires severe measurement, appearance, and formability (flow property, demold property), machine with injection speed and pressure control program is effective.

1.5 Backflow prevention ring

Backflow prevention ring is necessary for molding Lupital, since melt viscosity is relatively low.

If this backflow prevention ring is damaged by friction or corrosion, resin in the cylinder will backflow to the hopper side when injecting (pressure keeping), and might not give a injection pressure (hold pressure), described below, properly to the cavity because the cushion volume cannot be kept. Good molded product cannot be made in this case, so cushion volume and its stability when molding must be cared from the aspect of maintenance and control.

Backflow prevention ring do not need any special corrosion and abrasion resistant steel grade, but when molding glass fiber, carbon fiber filling grade filled filler, or other corrosive gas generating resin concurrently, usage of corrosion and abrasion resistant steel grade is preferable.
1.6 Peripheral equipment

1.6.1 Material drying

Iupital can be shape formed without preliminary drying the material, but if putting emphasis on mold deposit cut down, or surface appearance, 3 to 4 hours of material drying in 80 to 90°C is preferable. Especially, talking about each filler added complex Iupital, moisture absorption of filler might affect its physicality and appearance, so enforcement of preliminary drying is necessary.

Generally, shelf-type dryer or hopper dryer is sufficient for drying and there is no need to use dehumidifying dryer.

1.6.2 Mold temperature control

Iupital mold temperature control must have an ability to keep the mold temperature determinate by promptly getting rid of the heat brought by resin, to gain molded product with steady measurement and physicality. Table-1.6-1 indicates methods of the mold temperature control system.

<table>
<thead>
<tr>
<th>System</th>
<th>Adaptation to polyacetal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hydronic</td>
<td>Generally used to polyacetal. Be careful when removing scales in flow passage after long term usage.</td>
</tr>
<tr>
<td>Pressurized hydronic</td>
<td>Used when mold temperature is over 90°C</td>
</tr>
<tr>
<td>Heated oil circulation</td>
<td>Used when mold temperature is over 90°C</td>
</tr>
<tr>
<td>Heater calefaction</td>
<td>Irrelevant. Cannot stop mold temperature going to high. Sometimes used for auxiliary heating.</td>
</tr>
<tr>
<td>Chiller</td>
<td>Effective as a local overheat protector of the core and the other mold tool.</td>
</tr>
</tbody>
</table>

Select system by product required quality, workability, and mold design, but Heater calefaction is not suitable for general usage.

1.6.3 Local ventilation equipment

Iupital is a resin with a good thermal stability, but by the condition, it will generate formaldehyde by thermal decomposition.

Formaldehyde is a gas with a strong acridity against eyes, nose, and throat. Its exude can be noticed by the irritating smell. Even though there is a personal difference, this can be felt at 0.2 to 0.3ppm in general.

American Conference of Governmental Industrial Hygienists and Japan Industrial Hygiene Academic Society recommend "below 1ppm" in working atmosphere concentration. Consequently, formaldehyde concentration of the work floor such as around molding machine, should be "below 1ppm" and try to make it even lower.

For this reason, molding plant need to consider installation of part or general ventilation equipment. Also, formaldehyde detecting tube can be used for concentration measurement.
2. Molding condition

2.1 Preliminary drying

Figur-2.1-1 indicates relationship of drying temperature, drying time and water absorption rate of standard grade. Water absorption rate of Iupital is small, and molding without preliminary drying is possible if it is immediate after breaking the seal, but when putting emphasis on mold deposit cut down, or surface appearance, 3 to 4 hours of material drying in 80 to 90°C is preferable.

![Figure-2.1-1](image)

2.2 Resin temperature

Table-2.2-1 indicates standard preset temperature of cylinder in typical grade Iupital.

Because Iupital flow property resin temperature dependence is low, that flow length will not increase as expected even if resin temperature rises, and it can trigger thermal decomposition, which might cause mold deposit increase or silver streak.

Conversely, if the resin temperature is too low, Iupital crystal will be injected before plasticization meltdown, and molded product physicality might be insufficient, so preset temperature of at least 160°C to 170°C is necessary.

<table>
<thead>
<tr>
<th>Grade</th>
<th>Nozzle</th>
<th>Cylinder front section</th>
<th>Cylinder middle section</th>
<th>Cylinder rear section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard grade</td>
<td>180-210</td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>Weather-resistant grade</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>FC</td>
<td></td>
<td>200</td>
<td>190</td>
<td>180</td>
</tr>
<tr>
<td>FT</td>
<td></td>
<td>200</td>
<td>190</td>
<td>180</td>
</tr>
<tr>
<td>FL,FW</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>LO,FX</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>FA</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>FS</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>FU</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>ET</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
<tr>
<td>TC</td>
<td></td>
<td>190</td>
<td>180</td>
<td>170</td>
</tr>
</tbody>
</table>

- FS : Low screw rotation speed
- FU : Mold temperature below 40°C
- ET : Low injection speed
- Temperature of the upper limit 220°C
- Temperature of the lower limit 160°C
- Nozzle temperature adjust so that nozzle will not lock out or drool

2.3 Injection pressure

Injection pressure can be considered separately as fill pressure and hold pressure.

Generally, fill pressure should be set bigger than hold pressure.

In case of crystalline resin like Iupital, hold pressure is necessary for filling up because big shrinkage will happen when cooling solidification.

Increasing hold pressure is effective to resolve sink and void, but if it increases too much, it might cause burr.
2.4 Injection speed

Faster injection speed is preferable for thin molded product or multiple-cavity molded product with strict dimension accuracy. Conversely, slower injection speed is preferable for thick molded product.
Also, program control of injection speed is effective to resolve jetting and flow mark.

2.5 Mold temperature

60°C to 80°C for lupital mold temperature is generally suitable, and it is important point in molding condition.
Molding in mold temperature of about 20°C to 30°C by chiller is possible, for high-cycle molding purpose, but it might cause deformation from molded product residual strain, and dimension change from aftercontraction by the usage environment (high ambient temperature)
If measurement stability and surface gloss under high ambient temperature are necessary, setting mold temperature up to 120°C is effective.

2.6 Molding cycle

2.6.1 Molding cycle structure

General structure of molding cycle is as shown belos.

![Molding cycle diagram](image)

molding cycle \( \geq \) fill time + required solidification time + drying cycle time

2.6.2 Injection time (filling time, pressure keeping time)

Setting will vary by the molding machine, but think as below.

Injection time (filling time + pressure keeping time) \( \geq \) Gate sealing time

Gate sealing time is a time which resin stops solidification flow at the gate, and as Figure-2.6-2 shows, it can be thought as the injection time where molded product volume become constant when the injection time is longer.

If pressure keeping is stopped before gate sealing, melted lupital will backflow, which will end up with measurement and physicality varying widely. Warpage, sink, and void are caused from this in many cases.

To determine the gate sealing time, just measure injection time changed molded product volume.

![Gate sealing time](image)

2.6.3 Plasticization time

If the plasticization time is longer than required least cooling time, molding cycle will be longer at the same rate. In this case, raising screw rotation speed or cutting down plasticization time by using bigger plasticization volume molding machine is effective. Also, some molding machine can plasticize after mold opening by compounding performance.
2.6.4 Cooling time

Required solidification time is, minimum time that can extrude without molded product deforming after the pressure keeping time is over and Lupital is solidified. Of course, cooling time will differ by molded product thickness and draft angle, ejection system, location, hold pressure, and mold temperature.

Figure-2.6-3 indicates computed result of Lupital molded product thickness, mold temperature and required solidification time. Also, required solidification time in the molding cycle can be estimated from same figure, by calculating required solidification time of molded product thickest part, which is molding cycle rate controlling.

2.6.5 Drying cycle

Drying cycle is a total of mechanical time such as mold tool switching time, product ejection time, takeoff time, and intermediate time (waiting time). These time is to be controlled by molding machine model, product figure, mold system and structure, and differs by each molding condition.

2.7 Start and finish of molding operation, interruption of operation, and material change

2.7.1 Material change

When changing material in the heating cylinder, from Lupital to the other (material that might cause thermal decomposition by Lupital molding temperature, material that might decompose Lupital, or material which molding temperature differs widely), or the reverse, insert a polyethylene or polystyrene with wide molding temperature range, in the middle.

2.7.2 Interruption of operation • cleaning by dismantling

hen aborting the operation, keeping heating cylinder temperature under 150°C is preferable for security, and prevention from carbide mixing in.

Also, when aborting operation for long term, resin must be displaced, and polyethylene or polystyrene is preferable as a displacing material.

Through a long time molding operation, decomposed resin layer will be built up, and it will gradually carbonize inside the heating cylinder. This carbonization layer in the heating cylinder will grow thicker and if the decomposition become advanced, it will mix in the molded product, so occasional cleaning by dismantling is preferable.

2.8 Precautions

Avoid heating Lupital to more than 230°C, or remain Lupital in the molding machine cylinder of 200°C for long time. Like other polyacetal, Lupital will partially decomposed, and will extricate formaldehyde, if left for long time in high temperature above the melting point. Consequently, placing standard ventilation equipment is necessary at molding place.

If Lupital get overheated by mistake, lower the cylinder preset temperature, purge, put overheated Lupital in the water cistern directly, and try to prevent spreading gas to the operating environment.
3. Molded product quality

3.1 Dimension accuracy (tolerance)

SPI dimension tolerance table is helpful to see Lupital general dimension tolerance. SPI dimension tolerance table is indicated in Figure-3.1-1. In the case of multiple-cavity, measurement variability will be wider, compared to single-cavity.

![Figure-3.1-1 Lupital dimension tolerance](image)

3.2 Stable formability

To gain a stable dimension molded product, setting a proper molding condition is surely necessary, but in case of crystalline resin Lupital, temperature stability is especially important. This temperature include not only mold temperature and resin temperature, but also molding machine operating oil temperature and molding environment temperature.

It differs by the mold temperature, but basically Lupital needs approximately over 48 hours to stabilize from molding and shrinking. If want to determine whether the dimension is within the tolerance or not soon after molding, work out the dimension and the volume of condition changed molded product as preliminary experiment, and can simply control volume by measuring volume of molded product, soon after shape formed.
### 3.3 Mold defects and remedies

Table-3.3-1 indicates cause and remedy of mold defect phenomenon, which can be seen in lupital mold defect.

<table>
<thead>
<tr>
<th>Defect</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
</table>
| Silver streak                          | • Fluid in the pellet  
• Air engulfment when plasticizing  
• Resin decomposition by overheat (cylinder or nozzle part overheat, or existence of retention part) | • Drying pellet (80°C to 90°C for 3 to 4 hours)  
• Increase back pressure for better degassing when plasticizing  
• Lower the temperature where overheated  
• Clean the retention part or exchange to the parts with no retention part. |
| Tarnish or burn                         | • Resin decomposition by overheat  
• Too long residence time  
• Air engulfed by pellet | • Check cylinder, nozzle retention part, and fitting part.  
• Adjust cylinder temperature  
• Use molding machine with appropriate injection volume  
• Increase back pressure for better degassing |
| Partial tarnish or burn                 | • Heat production by Adiabatic compression from insufficient degassing in mold tool. | • Place air vent in the mold.  
• Lower the injection speed |
| Dark brown or black colored dot or small piece mixed in | • Peeled off decomposed resin film which was built inside the wall of the cylinder | • More cylinder purge cleaning before molding |
| Sink on the surface or bubble (void) inside | • Shrinkage is not filled by pressure keeping sufficiently, when plasticizing melted resin  
• Lack of material feeding | • Make holding pressure higher  
• Make the pressure keeping time loner (longer than gate sealing time)  
• Avoid nozzle block  
• Make thickness thinner  
• Make gate thicker  
• Place a gate at thick part  
• Confirm the cushion volume |
| Burr                                   | • Lack of mold locking force  
• Injection pressure too high  
• Injection speed too fast  
• wear of the mold  
• Resin melt viscosity too low | • Increase mold locking force  
• Lower the injection pressure and the holding pressure  
• Lower the injection speed  
• Fix or renew the mold  
• Change to high viscosity resin |
### Table 3.3-1(2) Cause and remedy of defect phenomenon

<table>
<thead>
<tr>
<th>Defect</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
</table>
| Demold defect or deformation when demolding | · Requires high demold power  
· Pressure reduction between the mold and the molded product  
· Demold power is not working at molded product and mold contact point  
· Molded product is not cooled enough when demolding | · Lower the injection pressure  
· Make draft angle  
· Polish the mold  
· Place device that avoids decompression at the mold  
· Make the mold opening slow enough  
· Increase ejector pin  
· Lower the mold temperature  
· Make cooling time longer |
| Lack of filling                             | · Cylinder temperature too low  
· Runner solidification too fast  
· Mold temperature too low  
· Uneven filling to each cavity  
· Lack of melted resin feeding volume  
· Degassing defection in the mold | · Raise cylinder temperature  
· Make runner size larger  
· Raise mold temperature  
· Change flow path, check up gate balance  
· Increase plasticization volume  
· Improve degassing effect within the mold (like gas vent) |
| Circular arc stripe and pock pattern at the edge part | · Low resin temperature  
· Low mold temperature  
· Low injection pressure and holding pressure  
· Slow injection speed  
· Lack of resin flowability | · Raise resin temperature, raise nozzle temperature  
· Raise mold temperature  
· Make injection pressure and holding pressure higher  
· Improve the injection speed  
· Improve the resin flowability |
| Flow mark (jetting, tarnish around the gate) | · By injected resin cooled down in the mold and solidified part swept again by melted resin | · Make the gate larger  
· Lower the injection speed  
· Change the gate location  
· Raise mold temperature and resin temperature  
· Improve resin flowability |
| Matter fleck                                | · Inappropriate melted resin flow  
· Drastic change in molded product section area  
· Inappropriate resin flow at the sharp corner | · Change section area not in a staircase pattern, and do it smoothly  
· Round out the sharp corner |
| Weld mark                                   | · Occurs when melted resin join together at flow end  
· Furthered by exhaust air defect in the mold | · Occurs when melted resin join together at flow end  
· Furthered by exhaust air defect in the mold |

### Table 3.3-1(3) Cause and remedy of defect phenomenon

<table>
<thead>
<tr>
<th>Defect</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
</table>
| Wrinkle around the gate                     | · Resin get cooled down before the pressure keeping, and design when filling get frosted | · Make the gate larger  
· Consider appropriate injection speed |
| Camber Deformation                          | · Uneven shrinkage of melted resin  
· Uneven molded product thickness  
· Uneven mold temperature  
· Uneven mold inner pressure  
· Anisotropy of molding shrinkage ratio | · Increase the injection pressure and holding pressure  
· Make consideration so that mold temperature equalizes (cooling ditch number and location)  
· Equalize molded product thickness, and symmetrize the design  
· Consider appropriate injection speed  
· Extend binding hour in the mold  
· Consider gate location and make gate section area larger |
4. Thermal stability

4.1 Thermal analysis

As an evaluation method of plastic thermal stability, there is TGA (Thermogravimetric Analysis) method, that examine volume decrease behavior caused by polymer thermal decomposition under high temperature melting condition.

Iupital good thermal stability can be seen in volume decrease behavior curve by TGA indicated in Figure-4.1-1.

![Volume decrease property by TGA (standard grade)](image)

Figure-4.1-1  Volume decrease property by TGA (standard grade)

4.2 Color change by retention

Iupital molding machine retention and color change property curve is indicated in Figure-4.2-1. Setting steady mold at resin temperature 190°C as standard, relationship of resin temperature and residence time in the case of $\Delta E=2.0$ is indicated.

![Molding machine retention color change property curve (F20-03)](image)

Figure-4.2-1  Molding machine retention color change property curve (F20-03)
4.3 Reproduction property

Basically, Lupital can be recycled when molding, by its good thermal stability.

Strength, molding shrinkage ratio, and color change, in regeneration rate 100%, 50%, 30% of standard grade Lupital is indicated in Figure 4.3-1 to Figure 4.3-4.

Even though repeated for 10 times in regeneration rate 100%, there was almost no physicality deterioration, but color tone and molding shrinkage ratio were changed slightly. There were no changes in strength, molding shrinkage ratio, and color tone, if recycled under 50%, and could say that it is keeping up the same performance with the virgin material.

However, there might be foreign material mixed inside, so still need a special attention. Also, mechanical property will decrease by fiber fracture, in filler filling grade, so recycling this is not preferable.

Tensile strength retention rate ... ○ 100% ■ 50% ▲ 30%

Figure-4.3-1 Change in tensile strength

Figure-4.3-2 Change in breaking elongation

Figure-4.3-3 Change in molding shrinkage ratio

Figure-4.3-4 Change in color
**4.4 Mold deposit**

Mold deposit (hereinafter called MD) is contaminated material attached to the mold when molding. In the case of polyacetal (hereinafter call POM) resin, if MD attachment become advanced, dimension accuracy cannot be gained. Also, it cause phenomenon like bad demolding and bad surface appearance.

**4.4.1 MD generation factor**

Following are some cause of MD
1. Added material to POM bled out and attached to the mold.
2. Formaldehyde [HCHO] gas polymerized again on mold surface, and generated paraformaldehyde.

**4.4.2 Molding condition and MD generation tendency**

There is tendency of MD generation and molding condition as the following shows, and should set condition reflecting these.

(1) Resin temperature
MD generation is fewer if it is lower.

| Table-4.4-1 Effect by molding condition (mold for MD measuring) : F20-03 | Relative value if the total of Molding condition 1 was 100 (Molding Machine IS90B) |
|-------------|-------------|-------------|
| Molding condition 1 | 10 | 90 | 100 |
| Molding condition 2 | 4 | 36 | 40 |

Cylinder temperature | Injection speed
Molding condition 1 | 230°C | 85%
Molding condition 2 | 195°C | 50%

(2) Injection speed
Do not make it too fast. (If injection speed is too fast, cavity air exhaust will not make it and cause adiabatic compression which will promote MD generation)

(3) Resin retention in cylinder
MD generation is fewer if it is shorter. Molding machine volume should avoid imbalance with product volume, and material retention within the cylinder.

(4) Mold temperature
MD generation is fewer if it is higher.

| Table-4.4-2 Effect by mold temperature (gear type MD evaluation type) : F20-03 undried | MD generated shot times |
|-------------|-------------|-------------|
| Mold temperature 25°C | from 400 |
| Mold temperature 80°C | 1400 OK |

Cylinder temperature: 230°C Injection speed: 85%

(5) Resin drying
MD generation will be promoted if wet, so dry resin sufficiently.

| Table-4.4-3 Preliminary drying effect : F20-02 | Formaldehyde gas and paraformaldehyde generated by molding |
|-------------|-------------|-------------|
| Solid material | Paraformaldehyde | Gas content | Formaldehyde gas |
| without drying (absorbed moisture approximately 0.4%) | 65 | 53 | 118 |
| with drying (absorbed moisture approximately 0.0%) | 15 | 85 | 100 |

(with drying relative value if the total was 100)
4.4.3 MD removal and caution about mold storing

(1) Clean the mold frequently.
   • If MD is only slightly attached, removing by just wiping with cloth is possible.

(2) Do not leave the mold with MD attached.
   • It will be a cause of rust especially in a hot and humid season from rainy season to autuminal rain.

(3) If MD is attached
   • If attached too much, remove by using bamboo spatula, copper, or brass which will not damage the mold.
   • Metal polish is also effective.
   • Use MD remover.

(4) MD remover
   By spraying at the start of MD attachment, it can prevent MD to be attached. If MD is attached excessively, removing will be ineffective.
5. Product • Mold designing

5.1 Product designing

Design of molded product is to make aimed product’s required performance more satisfactory, and need to be done after overall consideration of material practical physicality, formability, flowability and mold design condition.

Following are some basic point in molded product designing.

(1) Thickness to be equalized and make sure there is no drastic change in thickness (Figure-5.1-1)

Molded product thickness unevenness and drastic change in thickness will block resin flow and cause flow mark, camber by molding shrinkage ratio unevenness, and molded product deformation and strain by cooling speed unevenness.

(2) Make sure there is no undercut (Figure-5.1-2)

If there is undercut on molded product, demold problem will occur, and the cost will rise because the mold structure will become complex by placing slide core for demolding.

(3) Make sure there is no sharp corner (Figure-5.1-3)

Sharp corner blocks resin flow when molding, and will cause a flow mark. Also, talking about strength, notch effect and residual strain will occur which will likely to be a cause of strength deterioration.

(4) Consider draft angle (Figure-5.1-4)

Iupital is a material that has good slide property with the mold, but if the draft angle is insufficient, resistance will be big when demolding, which will cause deformation of molded product by ejector pin, so take it as big as possible.

(5) Thickness should not be excessively thick (Figure-5.1-5)

If the molded product thickness is excessively thick, defect phenomenon such as sink and void might occur. Also, it takes longer cooling time which will extend the molding cycle. If functionally necessary, place relief and try to make it even.

(6) Do not make the rib thickness excessively thick (Figure-5.1-6)

If thickness on a certain part cannot be relieved because of the strength, put a rib on and try to adjust thickness even.
(7) Molded product shape must be tough (Figure 5.1-7)

Figure 5.1-7

(8) Do not make the ratio of diameter and length of cylinder molded product too big. Long and thin core pin might fall over by the resin pressure when molding, or might cause problem such as crack. Also, center part of core pin will be relatively hot when molding, so inner diameter dimension variability might be wider and inner side skin layer might be ripped by resin inner pressure if molding cycle is short.

(9) Consider the ease of mold making
Make the mold shape that is easy to work and finish.

(10) Consider building up and fabricating for design

(11) Consider gate location and direction (Figure 5.1-8)
Consider weld location and resin orientation and determine gate location and gate number.

Figure 5.1-8

5.2 Mold designing

Injection forming mold holds important function which determines molded product productivity and quality performance, and must be designed so that required property such as molded product shape and cavity number can be exerted.

5.2.1 Cavity
Mold heating and cooling is especially important, and to control temperature and make temperature distribution better, there is need to separate in several parts for heating and cooling medium. Molded product is an article that is made by transcribing cavity, so must be well polished. Same push-out mechanism as normal synthetic resin is fine.

5.2.2 Structure
In the case of multiple cavity, dimension variability will be wide, so runner designing for multiple cavity simultaneous filling is important.
Family cavity, which shape form article with different cavity shape simultaneously, is basically not preferable.
Cavity location can be influenced by the gravity, and if it is double cavity, selecting horizontal location is preferable rather than vertical. Especially, in the case of thick molded product, there will be apparent jetting because of resin drooping to the bottom of the cavity by the gravity when filling, after getting through the gate.
In the case of multiple cavity, rather than (A) design in Figure 5.2-1, design with equal runner length is preferable like in (B) and (C).

Figure 5.2-1 Runner design
5.2.3 Parts structure

(1) Sprue · Runner

General slope will do for the sprue, but there must be no damages or bumps towards the demolding direction that would cause undercut. Figure-5.2-2 indicates runner cross-section shape. Basically, any shape is fine.

![Figure-5.2-2 Runner cross-section shape](image)

Cold slug well should be placed at sprue base and runner curve, to prevent cooled down resin flowing into the cavity.

Many types of hot runner is available in the market, but many of them cause resin retention problem, so installing after understanding each property is preferable. Especially, it is likely to cause resin retention in hot runner when talking about small molded product, so special care is necessary.

(2) Gate

It is important to consider molded product shape, cavity, performance, appearance, economic efficiency, and formability when selecting the gate. There are many types for gate.

(a) Direct sprue gate (Figure-5.2-3)

Used when it is single cavity, or want to directly attach the gate at the base of molded product. Injection pressure will directly reach molded product, so it is easy to generate residual strain, but have most simple mold design.

![Figure-5.2-3 Direct sprue gate](image)

(b) Side gate (Figure-5.2-4)

This type is most generally used, and it is rectangular or semicircular gate which can be attached to the side of the molded product, and used at multiple cavity mold well.

![Figure-5.2-4 Sidegate](image)

(c) Fan gate (Figure-5.2-5)

Its structure is similar to the side gate, but the gate width is spread like a fan, and used for big molded product.

![Figure-5.2-5 Fan gate](image)
(d) Pinpoint gate (Figure-5.2-6)
Pinpoint gate diameter is about 0.5mm to 2mm. Generally, there is no need for finishing. When talking about molding, gate seal is fast, and pressure will not directly be carried that residual strain will not be given to molded product. If the gate cross-section area is small, flow length will decrease and flow mark will likely to generate near the gate.

(e) Disc gate (Figure-5.2-7)
Used for preventing eccentricity and weld when molding disc-shaped or cylinder article. Have trouble with gate part finishing.

(f) Ring gate (Figure-5.2-8)
Used like the same as the disc gate, but weld will generate if ring gate shape is not considered like the ring part is first filled, then the cylinder part get filled.

Film gate (Figure-5.2-9)
Used in tabular molded product. Effective for preventing deforming by suppressing residual strain.

(h) Tab gate (Figure-5.2-10)
This is to place the tab at molded product side, and set up the gate there. Normally, the gate and the tab should be contacted in a right angle. Gate seal will happen at the gate, and residual strain and flow mark can be stopped inside the tab.
(1) Submarine gate (Figure-5.2-11)
This gate runner part is at the parting line side, but the gate will reach molded product side surface, through cavity plate or move type board, from runner part. Gate will automatically be cutoff when demolding molded product after mold opening.

(3) Change in thickness
(Figure-5.2-12)

(4) Draft angle (Figure-5.2-13)
Normally, 0.5° to 1° will be enough for molded product draft angle.

If grid, bigger draft angle is necessary. (Figure-5.2-14)

(5) Undercut (Figure-5.2-15)
Molded product undercut is essentially not preferable, but if it must be placed, make it below 2.5% of the diameter.

(6) Corner radius (Figure-5.2-16)
Corner radius of more than 1mm is preferable.
(7) Rib
Figure-5.2-17 will be the standard of ribbing.

(8) Boss
Figure-5.2-18 will be the standard of boss designing.

(9) Hole
See Figure-5.2-19 for hole designing, and be careful since molded product edge and holes are too close, the strength will be weaker.
There are many methods to put vent at the mold, but can be divided broadly into 3 methods.

- Degassing from mold division surface
- Degassing from cavity and core part
- Degassing by other special method

(a) Degassing from mold division surface (parting line)

In the case of POM, use 0.005mm to 0.02mm deep vent, and place as following.

1. As far place as possible from the gate
2. Where weld line likely to occur
3. End part of runner or sub runner

(b) Degassing from cavity and core part

1. Ejector pin using method
   
   Utilize ejector pin and its hole clearance. Pin and its clearance should generally be 0.02mm to 0.003mm for pin diameter of 5mm to 10mm, and for thinner pin, 0.01mm to 0.02mm is general.

2. Core pin using method
   
   If there is high boss or rib at the part of the product, degas by putting clearance around the core pin.

3. Layered nesting block method
   
   As a method of high rib degassing, make the thin block to layered nesting, and use that clearance. Also, insert a part of the cavity flat part as thin plate layered nesting, and degas from that thin plate clearance.

(c) Degassing by other special method

1. Degassing by vacuum suction
   
   Make the cavity elevation to the vacuum state by using the vacuum pump, and degas instantaneously. This method is almost an ideal method to degas, but the equipment cost will rise and mold structure will be complex. However, it also have transfer accuracy improvement effect against mold cavity too.

(11) Core pin falling

Core pin diameter and length ratio should be suppressed to 1:5 when one end is free end, because long and thin core pin might fall down from resin pressure. Furthermore, if both ends holding is possible, about 1:10 will be fine.
Mold temperature adjustment

Design of a mold cooling circuit is very important, since mold is thought as a resin heat exchanger when molding. Cooling hole diameter of normal Lupital mold should be more than $\Phi 8\text{mm}$, and less than $\Phi 12\text{mm}$.

Cooling hole location should be as near as possible to the cavity side, and make the hole clearance as small as possible. Shortest distance from cavity surface to cooling hole is determined by mold material, cooling hole dimension, shape, and cavity inner pressure.

Figure-5.2-24 indicates relationship of cooling hole dimension and distance from cavity surface to cooling hole. This curve calculates cooling hole diameter which is to cause $2\mu$ elastic change on cavity surface.

Distance between cooling holes should be 0.7 times of cooling hole diameter.

![Figure-5.2-24](image)

Distance $S$ from cavity surface to cooling hole (mm)

Cooling hole diameter $D$ (mm)

Figure-5.2-24  Cooling hole dimension and distance from cavity surface to cooling hole

However, placing like this is very difficult on actual mold, so it is structured by combination of various cooling circuit. There is straight, circle, multistage, spiral, flat turn, injection (bubbler tube), and separate plate (buffle plate), for cooling circuit category.

![Figure-5.2-25 to 28](image)

Figure-5.2-25  Straight cooling

Figure-5.2-26  Circle cooling

Figure-5.2-27  Multistage cooling

Figure-5.2-28  Spiral cooling

Figure-5.2-29  Flat turn cooling
Also, replacing mold material partially to the material with better thermal conductivity will take place, when placing cooling circuit is impossible. See Table-5.2-1 for mold material thermal conductivity.

<table>
<thead>
<tr>
<th>Mold material</th>
<th>Thermal conductivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carbon steel(S50C 0.5%C)</td>
<td>53</td>
</tr>
<tr>
<td>SKD61</td>
<td>34</td>
</tr>
<tr>
<td>Stainless steel(SUS304)</td>
<td>16</td>
</tr>
<tr>
<td>Zinc alloy(ZAS 4%AL,3%Cu)</td>
<td>109</td>
</tr>
<tr>
<td>Steel alloy (HR750 precipitation strengthened type : Kobe steel)</td>
<td>129</td>
</tr>
<tr>
<td>Beryllium steel 20C</td>
<td>121</td>
</tr>
<tr>
<td>Beryllium steel 275C</td>
<td>109</td>
</tr>
</tbody>
</table>
(13) Mold material

Following are some required property for injection molding mold.

(1) Tough and hard
(2) Good machinability
(3) Good grain and electrical spark machining
(4) Good mirror surface finishing property. Equal composition, no pinhole, and strong surface hardness.
(5) High corrosion resistance
(6) Good wear resistance
(7) Having thermal processability, and no big change by thermal process.

Characteristics shown above are required. Table 5.2-2 indicates typical mold material characteristics. See these characteristics and select mold material adapting the mold part requirement.

<table>
<thead>
<tr>
<th>Type</th>
<th>Hardness (HRC)</th>
<th>JIS type</th>
<th>Machinability</th>
<th>Specularity</th>
<th>Grain finishing</th>
<th>Corrosion resistance</th>
<th>Wear resistance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-hardened steel</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>14</td>
<td>S55C</td>
<td>10</td>
<td>4</td>
<td>6</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>27</td>
<td>SCM445</td>
<td>8</td>
<td>6</td>
<td>7</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>33</td>
<td>Upgraded SCM</td>
<td>9</td>
<td>5</td>
<td>6</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>Upgraded SNMG</td>
<td>8</td>
<td>9</td>
<td>9</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Deposition hardened free-machining</td>
<td>8</td>
<td>8</td>
<td>7</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Deposition hardened type</td>
<td>7</td>
<td>9</td>
<td>10</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>35,40</td>
<td>Upgraded SUS630</td>
<td>3</td>
<td>8</td>
<td>8</td>
<td>10</td>
<td>4</td>
</tr>
<tr>
<td>Hardening steel</td>
<td>53</td>
<td>Upgraded SUS420J2</td>
<td>7</td>
<td>10</td>
<td>9</td>
<td>9</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>42–52</td>
<td>SKD61</td>
<td>8</td>
<td>8</td>
<td>7</td>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>55–60</td>
<td>Upgraded SKD11</td>
<td>6</td>
<td>9</td>
<td>8</td>
<td>6</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SKD11</td>
<td>5</td>
<td>7</td>
<td>5</td>
<td>6</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Upgraded SUS420</td>
<td>7</td>
<td>7</td>
<td>7</td>
<td>5</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Upgraded SUS440C</td>
<td>5</td>
<td>9</td>
<td>8</td>
<td>8</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>63–68</td>
<td>Deposition hardening steel</td>
<td>5</td>
<td>8</td>
<td>7</td>
<td>6</td>
<td>10</td>
</tr>
<tr>
<td>Powdered high-speed</td>
<td>55</td>
<td>Maraging steel</td>
<td>4</td>
<td>10</td>
<td>10</td>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>Nonmagnetic steel</td>
<td>2</td>
<td>8</td>
<td>–</td>
<td>6</td>
<td>7</td>
</tr>
</tbody>
</table>

Attention: Bigger the number, better in the table
6. Flow property

6.1 Flow property of Iupital and other resin

Flow property of each grade Iupital and other thermal plasticity resin is indicated in Figure 6.1-1.
6.3 Molding condition and specific flow length

Flow length indicates various tendency by the molding process condition. Secondly, relationship of molding process condition and flow length is indicated as specific flow length. Following are some standard of specific flow length calculating condition.

Grade: F20-03
Injection pressure: 79MPa(800kgf/cm²) Mold temperature: 80°C
Resin temperature: 200°C Injection speed: 68cc/sec

Say, flow length of this condition is L0, and flow length of each condition is L, which mean specific flow length will be L/L0. As shown below, specific flow length indicates linear relationship with injection pressure and resin temperature, and curve relationship with mold temperature. Also, talking about each grade lupital, relationship of each grade gate diameter and specific flow length can be put on each curve.

6.3.1 Injection pressure

Relationship of injection pressure and specific flow length is indicated in Figure-6.3-1.

![Figure-6.3-1 Injection pressure and specific flow length](image)

6.3.2 Mold temperature

Relationship of mold temperature and specific flow length is indicated in Figure-6.3-2.

![Figure-6.3-2 Mold temperature and specific flow length](image)
6.3.3 Resin temperature

Relationship of resin temperature and specific flow length is indicated in Figure-6.3-3.

![Resin temperature and specific flow length](image)

6.3.4 Gate diameter

Relationship of gate diameter and specific flow length is indicated in Figure-6.3-4.

![Gate diameter and specific flow length](image)
6.4 Flow value (MI) and flow length

Iupital flow value can be indicated in measurement value (MI value: g/10min) by melt indexer. Relationship of this MI value and flow length is indicated in Figure 6.4-1.

Also, \( \log\text{MI} = K \cdot (\text{L}/\text{tn}) \) \( \cdots \cdots \) (1) formula

- \( K \) : Coefficient
- \( L \) : Flow length
- \( t \) : Thickness

\( n : F10=1.4 \)

\( F20, F30=1.5 \)

\( F40=1.6 \)

Figure 6.4-2 relationship is indicated by arranging Figure 6.4-1 relationship as shown above. Coefficient \( K \) can be calculated from material MI value in the same figure, and flow length can be estimated by using (1) formula relationship.
7. Shrinkage characteristics

7.1 Molding shrinkage

As a cause of molding shrinkage, expansion shrinkage by heat, compressive elasticity recovery, specific volume change by crystallinity change, and molecular orientation, are thinkable. See Figure-7.1-1 for relationship of standard grade Iupital specific volume and temperature. Iupital, crystalline resin, molding shrinkage will change greatly by the factor change shown above, so understanding potential shrinkage is important when mold designing.

Molding shrinkage will be affected by molded product shape (thickness), molding condition (injection pressure, injection time, mold temperature, and resin temperature), and mold design (sprue and runner size, and gate shape).

![Figure-7.1-1 Relationship of specific volume and temperature](image1)

**Figure-7.1-1**  Relationship of specific volume and temperature

**7.1.1 Thickness and molding shrinkage**

Molding shrinkage will change greatly by molded product thickness, and smallest value will be indicated when thickness is 2mm to 3mm if the molding condition and molded product shape is the same and only thickness changes. Relationship of standard grade Iupital molded product thickness and molding shrinkage ratio is indicated in Figure -7.1-2.

![Figure-7.1-2 Molded product thickness and molding shrinkage ratio](image2)

**Figure-7.1-2**  Molded product thickness and molding shrinkage ratio
7.1.2 MI value and molding shrinkage

Relationship of standard grade Iupital MI value change and molding shrinkage ratio is indicated in Figure 7.1-3.

![Figure 7.1-3](image)

7.1.3 Molding shrinkage ratio and anisotropy

Difference between machine direction and that transverse direction of molding shrinkage is called anisotropy of molding shrinkage ratio. Talking about standard grade Iupital, anisotropy of transverse direction shrinkage subtracted from machine direction shrinkage is almost “0”. This anisotropy sometimes indicates large value by the grade, so need a special attention. Anisotropy of each grade Iupital is indicated in Table 7.1-1.”

Table 7.1-1 Shrinkage ratio anisotropy of each grade Iupital Unit: %

<table>
<thead>
<tr>
<th>Grade</th>
<th>Machine direction</th>
<th>Transverse direction</th>
<th>Anisotropy</th>
</tr>
</thead>
<tbody>
<tr>
<td>F10</td>
<td>1.8</td>
<td>1.8</td>
<td>0.0</td>
</tr>
<tr>
<td>F20</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>F30</td>
<td>1.5</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>F40</td>
<td>1.5</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>FK10-01</td>
<td>2.0</td>
<td>2.0</td>
<td>0.0</td>
</tr>
<tr>
<td>FV-30</td>
<td>1.5</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>FG1025A</td>
<td>0.6</td>
<td>0.9</td>
<td>-0.3</td>
</tr>
<tr>
<td>FG2025</td>
<td>0.5</td>
<td>0.8</td>
<td>-0.3</td>
</tr>
<tr>
<td>MF3020</td>
<td>1.5</td>
<td>1.0</td>
<td>0.5</td>
</tr>
<tr>
<td>FB2025</td>
<td>1.3</td>
<td>1.3</td>
<td>0.0</td>
</tr>
<tr>
<td>FC2020D</td>
<td>0.2</td>
<td>0.6</td>
<td>-0.4</td>
</tr>
<tr>
<td>FC2020H</td>
<td>0.3</td>
<td>0.6</td>
<td>-0.3</td>
</tr>
<tr>
<td>FT2010</td>
<td>1.4</td>
<td>1.4</td>
<td>0.0</td>
</tr>
<tr>
<td>FT2020</td>
<td>0.8</td>
<td>0.8</td>
<td>0.0</td>
</tr>
<tr>
<td>LO-21</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FX-11</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FL2010</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FL2020</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FW-21</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FW-24</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FM2020</td>
<td>1.6</td>
<td>1.6</td>
<td>0.0</td>
</tr>
<tr>
<td>FU2025</td>
<td>1.3</td>
<td>1.3</td>
<td>0.0</td>
</tr>
<tr>
<td>FU2050</td>
<td>0.8</td>
<td>0.8</td>
<td>0.0</td>
</tr>
<tr>
<td>ET-20</td>
<td>1.1</td>
<td>1.1</td>
<td>0.0</td>
</tr>
<tr>
<td>TC3015</td>
<td>1.2</td>
<td>1.2</td>
<td>0.0</td>
</tr>
<tr>
<td>TC3030</td>
<td>0.9</td>
<td>0.9</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Ph: 78.5(MPa) (800k gf/cm²)
Tm: 80°C
Tc: 200°C
4in circular plate, 3mm
7.1.4 Molding condition and molding shrinkage

Iupital molding shrinkage ratio will be affected and change by molding condition too. Following indicate tendencies of molding conditions that especially give an impact to Iupital molding shrinkage ratio.

(1) Injection pressure (holding pressure)
   As indicated in Figure-7.1-4, Iupical molding shrinkage ratio and injection pressure (holding pressure) shows inverse proportion relationship.

![Figure-7.1-4](image)

Figure-7.1-4 Relationship of molding shrinkage ratio and holding pressure (standard grade)

(2) Injection time
   As indicated in Figure-7.1-5, Iupical molding shrinkage ratio and injection time shows direct proportion relationship until the gate plasticize, but after the gate plasticization, molding shrinkage indicates almost constant value.

![Figure-7.1-5](image)

Figure-7.1-5 Relationship of molding shrinkage ratio and injection time (standard grade)
(3) Mold temperature

As indicated in Figure-7.1-6, Iupical molding shrinkage ratio and mold temperature shows direct proportion relationship.

![Graph showing the relationship between mold temperature and molding shrinkage ratio.](image)

*(Figure-7.1-6 Relationship of molding shrinkage ratio and mold temperature (standard grade)*)

(4) Resin temperature

Relationship of Iupital molding shrinkage ratio and resin temperature is as shown in Figure-7.1-7, but compared to mold temperature, it impact is small.

![Graph showing the relationship between resin temperature and molding shrinkage ratio.](image)

*(Figure-7.1-7 Relationship of molding shrinkage and resin temperature (standard grade)*)
7.2 Thermal shrinkage and dimension change by thermal process

Talking about crystalline resin lupital, recrystallization will be promoted by annealing or heating under actual usage condition and will cause dimensional shrinkage. Molded product dimension will change by its molding condition, heating temperature, and heating time, so special attention is necessary. Figure 7.2-1 to Figure 7.2-6 indicate thermal shrinkage data of standard grade lupital of molded product thickness 1mm to 3mm, mold temperature 50°C and 80°C, heating temperature 60°C to 150°C, 30min to 120min heating time. Test piece dimension is 70mmL*20mmW*tmm.

Effect of heating temperature and time to thermal shrinkage ratio

![Figure 7.2-1](image1)

![Figure 7.2-2](image2)

Effect of molded product thickness to thermal shrinkage ratio

![Figure 7.2-3](image3)

![Figure 7.2-4](image4)
7.3 Aftercontraction Molded product dimension change by time elapsed

Molded product will cause dimensional shrinkage with the progression of crystallization soon after molding. Especially, it will dimension change for long time when the molding temperature is low when molding. Figure-7.3-1 indicates lupital long-term dimension change data.

(long-term shrinking change by Φ 30 circular plate soon after molding)