Design Guide

R&D Center

KOREA ENGINEERING PLASTICS CO.,LTD.

1. Shape design of product

(1) Basic Principles

One of the most important parts of tool design for injection molding is to keep a consistent wall thickness of the product and as such every design must focus on stabilizing wall thickness. This minimizes the troubles resulting from contraction during the injection molding process. Therefore, it is important to remove any thick parts in the product while designing plastic products. When deciding the basic thickness of the product, the designer should consider not only the structure, function, and appearance, but also the influence on injection molding effectiveness (injection pressure for tool stopping, cooling time, and extraction convenience from the tool). Moreover, injection molding pressure level, speed, stiffness of the tool, and the resin's flow path.

The common ground among the 4 factors to succeed at plastic product development (resin, product design, tool design, and molding) is CAE (Computer Aided Engineering). A developer can take advantage of CAE by simulating and evaluating the resin for its product and tool design. However, the key to success is to understand its limits and utilize the core value from it.

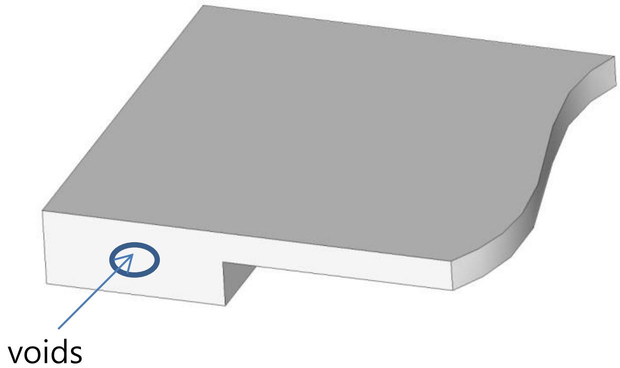
The main principals during product design are as follows:

1) Provide a draft
   Provide enough draft for easier release
2) Consistent thickness of the wall
   - Shrinkage problems occur when the wall is too thick, while cracks can be observed if it is too thin
   - Minimum ± 20% change in the wall thickness is preferred
3) A simple shape
   A simple symmetrical (mold shrinkage is consistent) shape is recommended. A parting line where the mold is split stays simple (flat surface on a line)
4) Avoid sharp corners
   Sharp corners cause cracks by concentrating stress

(2) Wall Thickness

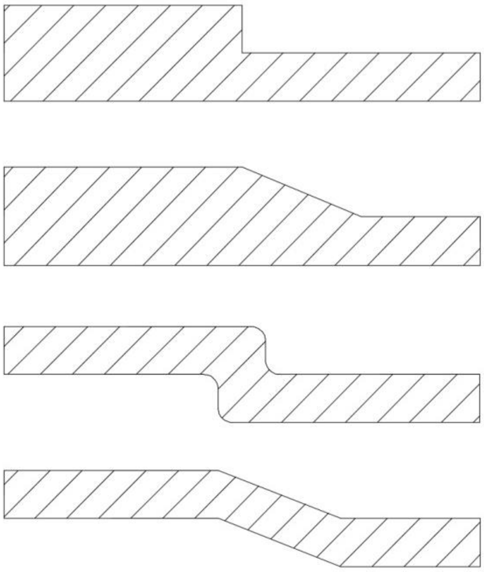
Wall thickness is the prime factor that determines cycle time in injection molding. Thus, minimizing wall thickness is the most important part in plastic product design as thinner wall thickness can reduce material usage and production time. Material cost and cooling time comprise 70% of product cost and directly relates to economic feasibility.

Wall thickness has to be as consistent as possible to avoid partial distortion, inner stress, and cracks. If it is not, voids can be observed in the thick area due to shrinkage.

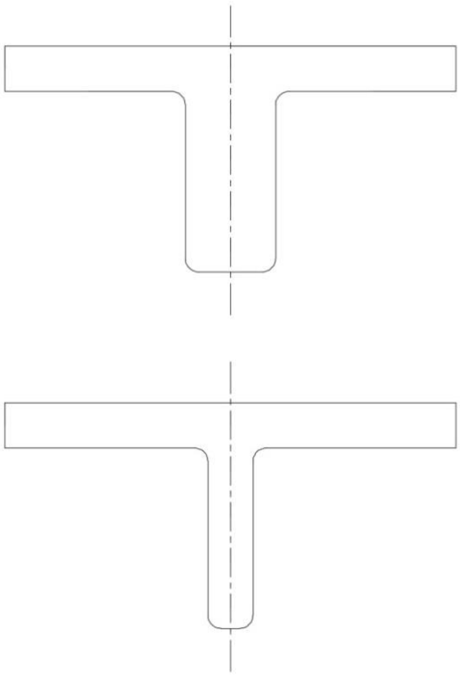


[Figure. 1] Wall thickness design example (minimize variation by gradual change)

If it is not possible to keep wall thickness even, then minimize the variation of wall thickness and change it gradually. Changes of wall thickness can cause negative influences such as shrinkage fluctuation, flection, and residual stress.



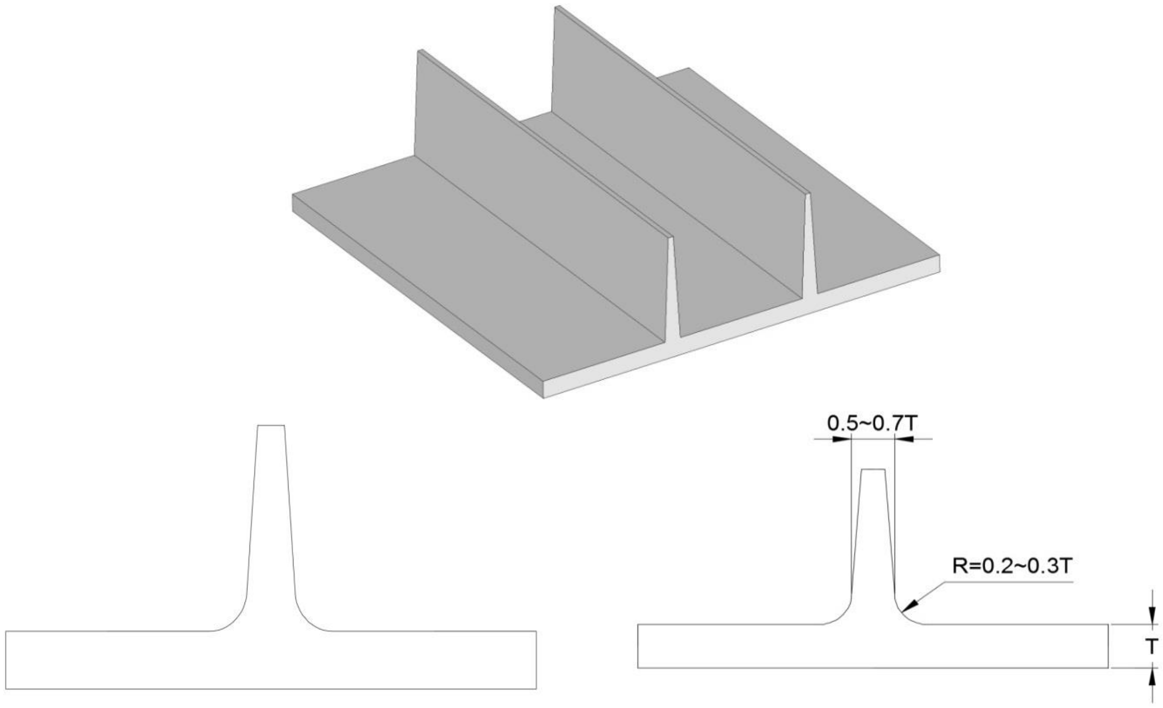
[Figure. 2] Wall thickness design example (minimize the variation by gradual change)



[Figure. 3] Wall thickness design example with ribbing

(3) Rib

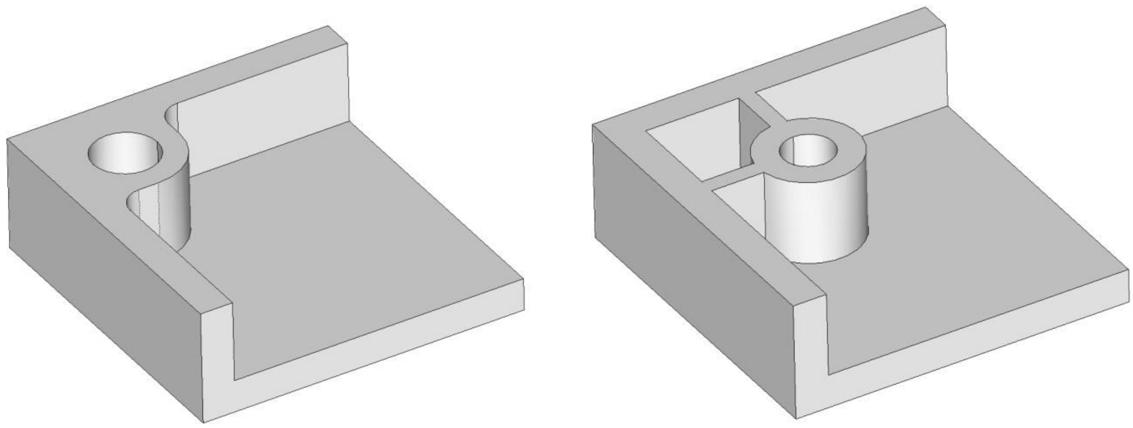
Ribs can be used to increase the strength of the product. A thicker wall increases the strength of the product but also increases the cost. To avoid it, ribs can help the product to meet the target strength under the same wall thickness. Keep the ribs as short as possible and design the product to have symmetrical drafts, which help with releasing. Short ribs prevent troubles during stopping and avoid excessively narrowed tips by the draft. For this reason, multiple short ribs are more preferable than a single long rib.



[Figure. 4] An example of effective rib design

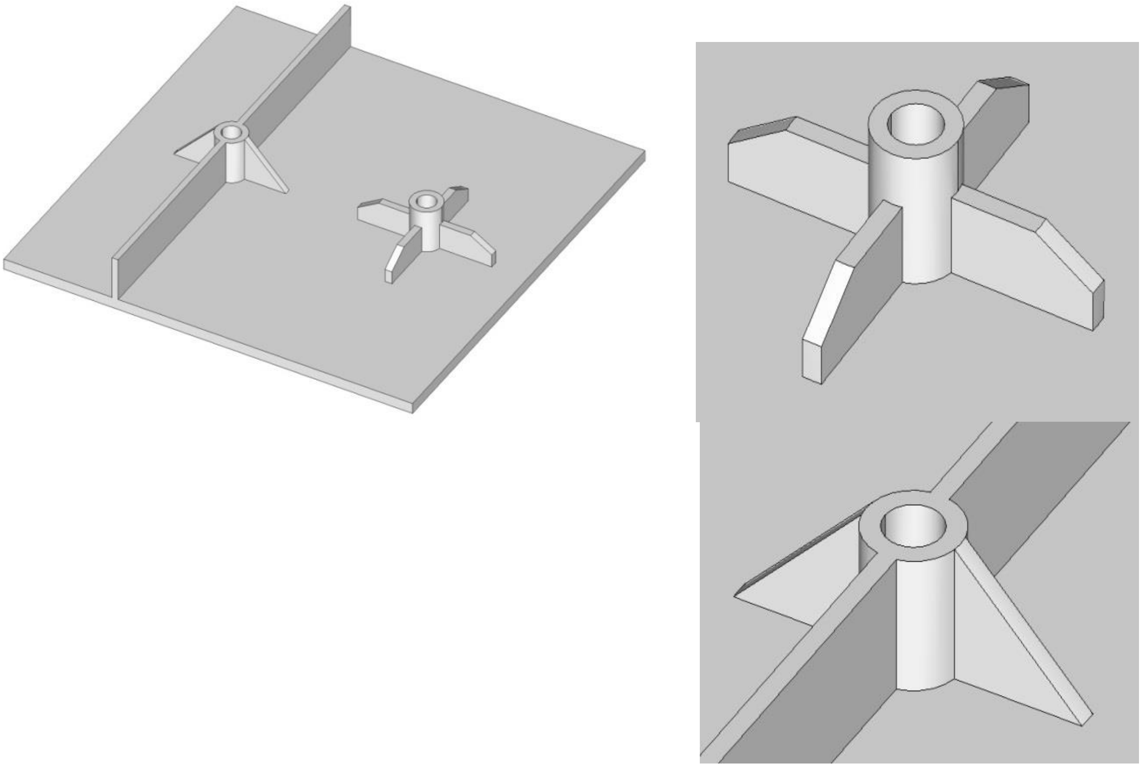
(4) Boss

A boss is a projected hollow or solid circle shape in the basic wall of the product. The purpose of a boss is to assemble one part with others by a self-tapping screw, extended insert, stationary fit plug, and drive pin. A boss is connected or attached to the side wall by the ribs. If the boss is directly attached to the side wall, the side wall gets thicker and sink-marks or voids can occur easily. A hollow boss can provoke weld lines depending on the material flow characteristics around the core. Internal diameter is decided by the function of the boss, for example a self-tapping screw's pitch diameter is the boss's internal diameter. A boss's external diameter shall be about 2.5 times, which fulfills both decent strength and thin thickness.



Poor Design (Sink)　　　Optimal Design

[Figure. 5] Boss design example



[Figure. 6] Efficient design example

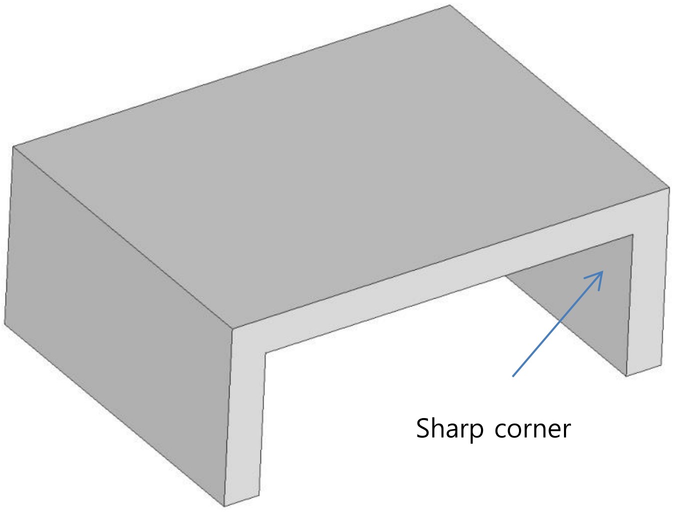
(5) Core

Sometimes the product needs a long core like the inner part of the boss or itself shaped as a long cylinder. Those with long cores have several possible problems. When under high pressure, the core can be inclined and wall thickness can be uneven or the stopping pattern could look different from what it is expected. Those problems can be solved by limiting the highest core height to be a maximum of 2.5 times then the core’s diameter.

A core incline problem can occur when installing a gate in the center of the core to help material flow into the cavity symmetrically. There are 2 ways to reduce core flexure: by using the self-centering core or making a thicker bottom of the cup-shaped product compared to its side wall.

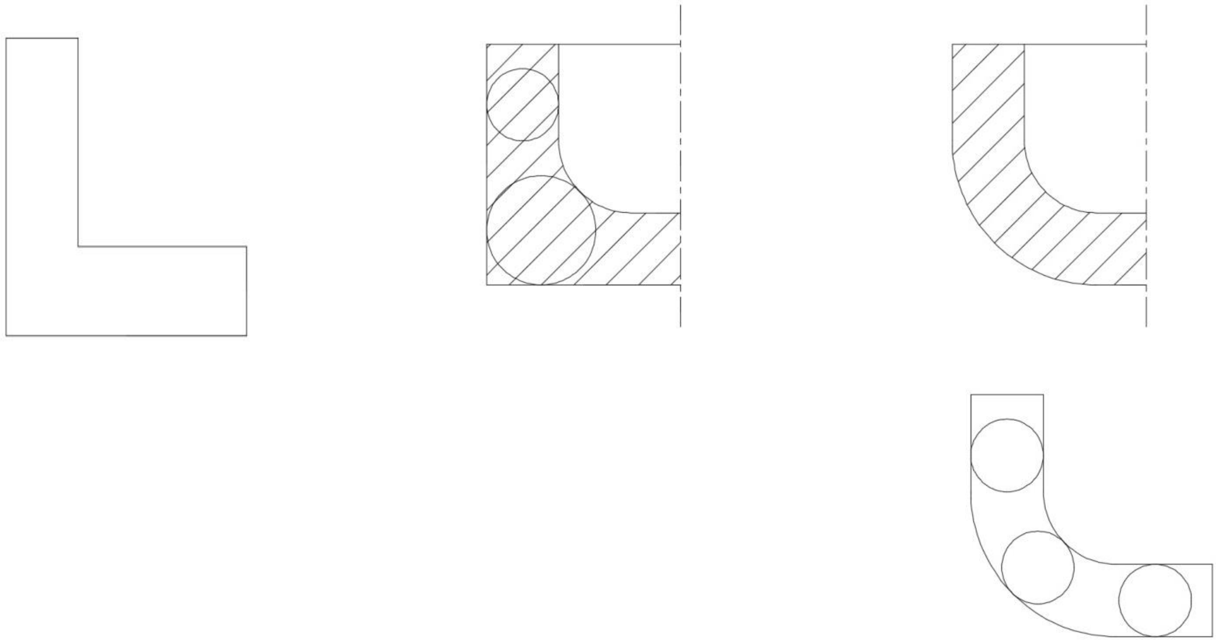
(6) Fillet and Radius Section

Most plastics are very sensitive to notching, so it is better to avoid sharp edges. Sharp corners from the product generate molding stress and can be easily break when exposed to external stress. In particular, notches from internal edges become a source of stress concentration.

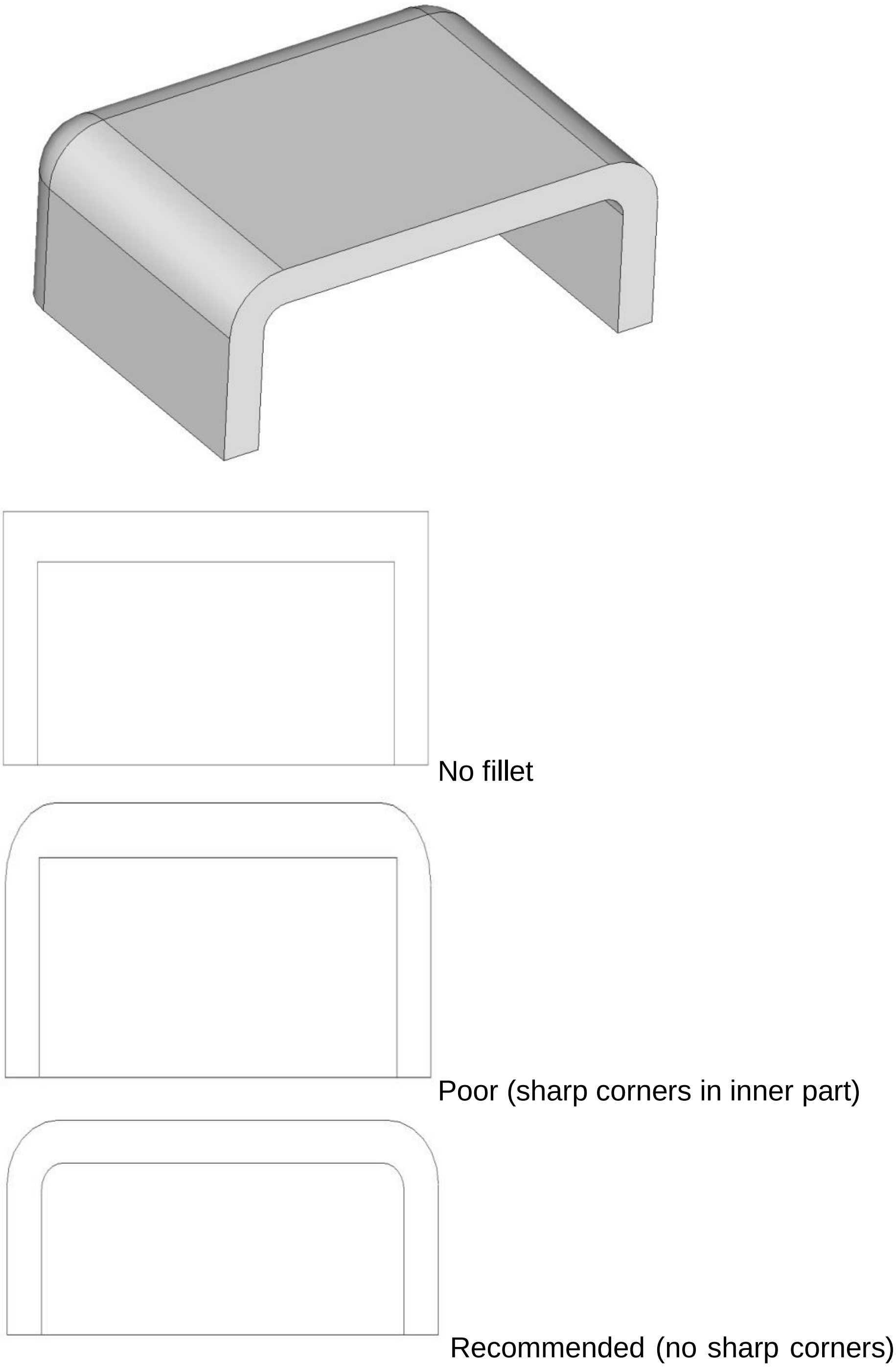


[Figure. 7] Sharp corner example in molding product

Stress concentration can be relieved by putting fillet proportionate the thickness of the corner. The minimum radius ratio upon the wall thickness is 0.5. Sharp parts, of course, are vulnerable to load. Some parts which form an edge with the main part (e.g. Ribs, gusset, boss, etc.) need to have 2 times more R/T then the recommended R/T. Thick parts where 2 sides meet together must be as thin as possible. To maintain the thickness even in the corner part, the inner radius has to be 50% bigger than the main part and outer radius size has to be added to the value of the inner radius and wall thickness(a larger inner radius is recommended).



[Figure. 8] Outside corner design example

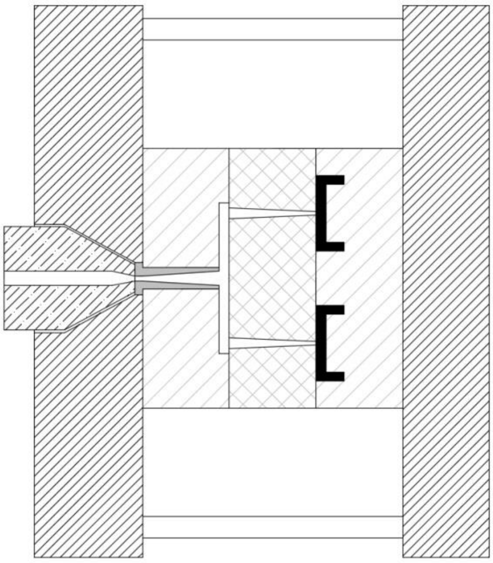


[Figure. 9] Comparison of fillets

2. Mold Design

The minimum requirement of mold design is that the stopping has to be done by designated plastics, has to be strong enough to bear internal and external force, and the product must be ejected. In other words, mold design shall eliminate any factors that prevent the product from being ejected from the tool.

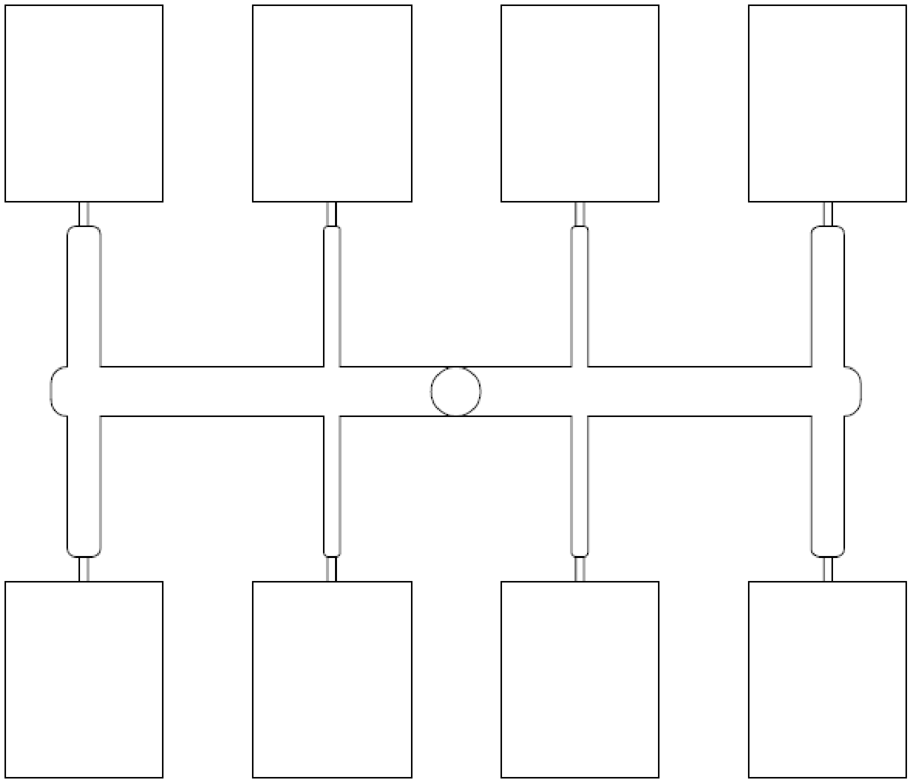
The most important principals while designing the tool for injection molding are as follows:
- Increase the tool's stiffness such that it can be free from any partial disposition or distortion
- Consider ejection
  Provide sufficient draft and polish
- Install air vent (gas out)
  Install air vent (gas vent) at the point
- Improve cooling efficiency
  Install cooling water holes with sufficient number and size to circulate sufficient cooling (or heating) water in the tool
- Forecast flowage
  Forecast the flowing pattern of the material and set the position of the weld line



[Figure. 10] General mold structure for injection molding

(1) Cold Runner

The runner is one of the most important factors during injection molding as it delivers the material from sprue to the gate. The tool designer must consider a structure which avoids melted material from reducing injection force by virtue of its own flow resistivity as it flows into the runner to this end, flow resistivity absolutely must be mitigated. The design also needs to consider the cavities to be filled at the same time. Flow channel efficiency can be maximized by minimizing the ratio between superficial dimension and cross sectional areas. A perfectly circled runner is most recommended, while half a circle and trapezoidal cross section is also fine. In most cases, a runner's diameter is better to be thicker than the wall thickness at the gate. Shorter runners can minimize the re-processing time and reduce the loss of pressure. In principal, runner arrangement has to be designed so that each cavity has the same pressure. In other words, by changing the runner and sub-runner's diameter, the filled volume of each cavity can be controlled. In addition, if a different runner's length affects the product shape, it can be improved by changing the second runner's depth and length to fill the multiple cavities to be filled at the same time.



[Figure. 11] Example of modified sub-runner's diameter for evenly filled cavities

Principles of Runner Design

A runner system in the tool is to lead melted materials to an empty space (cavity), so that the final product can be produced. Runner arrangement, size, and connection form affect the filling process and ultimately affect the product's quality. Economic efficiency (short freezing and cycle time) centered design can be in contrast to the product's quality. The principles of runner design are as follows:

1) Each cavity has to be fully filled at the same moment when the last tip of the flowing part is completely filled.
2) Design the tool with a shape and layout which causes less flow resistivity. In addition, a short runner is preferable to minimize lost pressure, temperature, and material.
3) Avoid drastic pressure loss in the runner.
4) Runner thickness has to be thicker than the product until the product gets solid enough to deliver the holding pressure.
5) In principal, a multi-cavity tool has to have the same runner size and the shape. An especially accurate product has to follow this rule and be handled precisely.
6) In terms of pressure delivery, larger cross-sectional areas are preferred.
   The outer line's length should be minimized to produce a better thermal conduction.
7) 50 rms is fine for the runner's surface as same as the cavity. This reduces pressure drop and assists runner's separation.

8) Generally, only one part of the mold is processed for easier runner modification. In this case, design a trapezoidal runner.

Runner Size

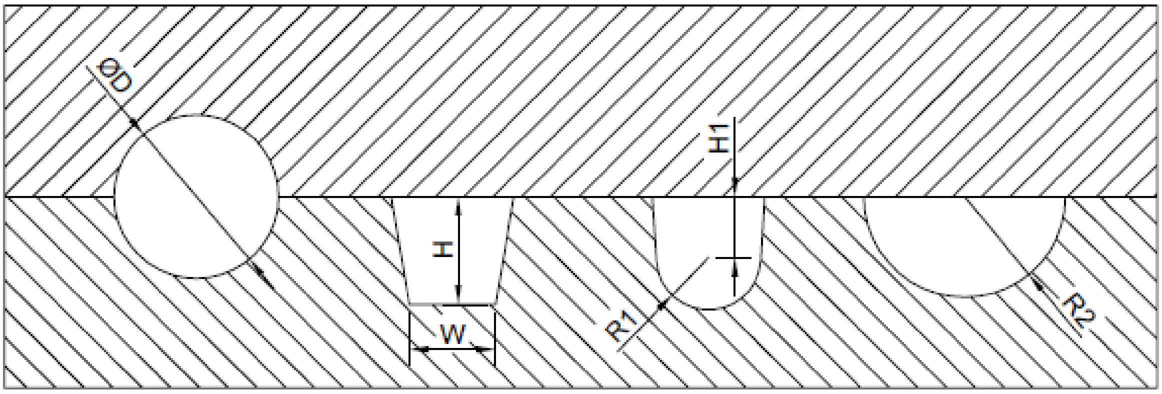
The runner has to be designed so that material with the same melt index shall be filled simultaneously as the others under same conditions (temperature, pressure). At the same time, it has to fulfill the basic requirement of the least pressure loss and material consumption, which is a diametrically opposed idea. The runner itself is not only a path for melted material but a path to deliver holding pressure. For this reason, in general, it has to be at least 1.5 mm thicker than the product's thickest part to deliver holding pressure effectively.

Runner diameter > S_{max} + 1.5 ㎜

S_{max} : the thickest part in the product

Runner Cross Section Shape

The circle cross section is the most recommended one as it has the least flow resistivity among the same cross section size. However, due to easier ejection, semicircle and trapezoidal shapes are also used.



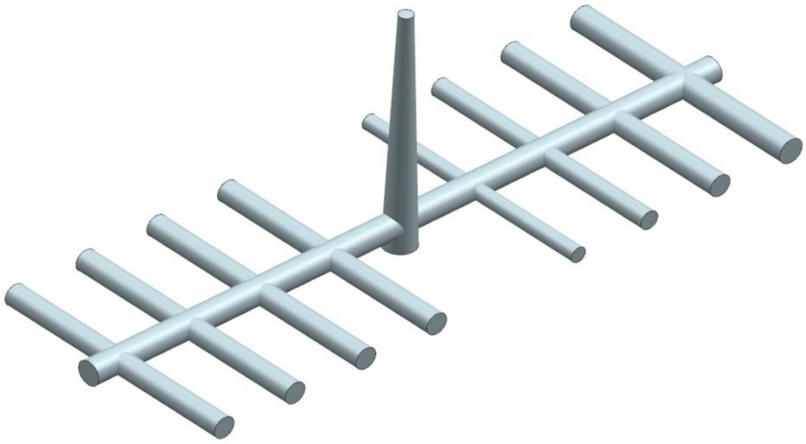
[Figure. 12] General runner's cross section example

Runner Layout

A multi-cavity tool's cavity arrangement depends on product shape, number of gates, plate composition, and gate figure. Runner layout design must follow these factors:

1) Runner design has to be the shortest possible to avoid pressure and material temperature loss.
2) Consider the runner balance to have cavities fully filled simultaneously.
3) If there are multiple gates, the runner has to be set considering the gate balance.
4) To prevent inflow of any cold slug material into the cavity, thoroughly process the cold slug at the tip of the runner.

Even though the runner is designed geometrically, because of asymmetrical heat from the runner, cavity by cavity's fill-up time, temperature, and pressure dispersion might be different. Geometrically-balanced runner designs can be improved by CAE analysis to modify each cavity's 2nd or 3rd runner's diameter to control the runner's diving point.



[Figure. 13] Example of runner system layout

(2) Hot Runner Mold

A hot runner keeps the material in a melted condition from every flowing part so that there is no sprue runner left at the end of the process and automation efficiency improves. In the hot runner mold, material stays a melted and does not eject during the molding cycle. Compared to a cold runner mold, a hot runner has the advantage of not making or scrapping the runner as the material is filled and only the final product is ejected. Without a runner, injection volume, plastication time, runner cooling time, and tool opening stroke are reduced. Clamping force also decreases as there is no injection power like in a cold runner mold. In any case, through those advantages, a smaller capacity machine can mold the product. Hot runner design is as similar as a 3-plate cold runner mold design which has a flexibility of having gate position by putting runner path on the upper part of the cavity. As it can use a thick flow channel unlike the cold runner system, it can install the gate free from pressure drop, which occurs in cold runner systems.

The advantages of a hot runner

1) It can minimize loss from sprue and runner.

2) It can save time for sprue and runner filling and cooling time, and less metering volume, short clamp stroke needed to take out the sprue and runner gets shorter so that cycle time is reduced.

3) Metering volume decreases and lower injection pressure and clamping force provides a wider range of options for selecting the molding machine.
4) Each gate's temperature is controlled by the hot runner and every material coming through the each gate has a consistent temperature. When the valve gate is used, each cavity has the same holding pressure and this minimizes quality variation.

5) Even large product can be produced without weld lines by using valve gates to control the gate. In general, a hot runner can reduce or eliminate weld lines with less gate compared to cold runners.

Weakness of the hot runner

1) Parts, such as controller (electrical, hydraulic controller for valve gate) and manifold are more expensive than cold ones and it also has extra equipment which can cause malfunctions.
2) Mold modification such as gate position change, manifold diameter change, and increasing number of gates are difficult so it needs careful review during the design.
3) Using multiple materials or changing color is difficult.

4) Manifold is in the fixed part, which brings difficulty in locating the cooling line.
5) Material that is sensitive to heat, high content G/F reinforced material, high performance material with a high melting point has a limit to use or to find a proper type of hot runner system.
6) Always in melted condition and leakage can happen as many parts are assembled.

Hot runner's heating method

A hot runner is composed of manifold, drop, and nozzle. Manifold and drop has a heater in or outside to keep the material melted. As for POM, heating from outside of both manifold and drops are recommended to avoid thermal decomposition and formaldehyde gas emission.

When both manifold and drops are placed outside, it has a lower pressure drop in the hot runner and color change is easy. However, it can have leakage problems between manifold and drop, and deformation can occur when the heater at the outside does not effectively cool down the fixed part of the mold. When the heater is in both manifold and drop, even though it has less leakage trouble, it leads the material's decomposition where heaters are intersected.

(3) Gate

The gate is the pathway that connects cavities and runners and affects product function, formability, productivity, and workability. Gate design is one of the most important aspects of tool design as so many troubles can arise through improper gate position, number of gates, types, and sizes.

Gate's function and role

The gate delivers the holding pressure to the cavities to ensure the product's dimension and dimensional stability. It also controls the material and fiber-type reinforcement's orientation that affects deformation. Depending on size and the location, it controls the injection volume of the material (by that, weld line location can be modified) and improves the materials' fluidity from the frictional heat occurred by the pressure drop (reheating material). The weld line's location, numbers, and stiffness are directly affected by the gate's location and numbers.

Factors for consideration during the gate design

The gate's position, number, shape, and its size are the main factors to be considered.
The gate's position affects deformation, sink marks, voids, dimension, weld line location, jetting, short shot, injection balance, and orientation. The number of weld line and its position, injection pressure, clamping pressure, machine specification, and circularity should be considered, as well. The gate's shape has to be determined by its automation availability, gate marks on the surface, product shape, and size. Gate size affects the holding pressure time and influence on dimension, sink marks, voids, and deformation. Furthermore, increasing material temperature by excessive shear heat and discoloration, gas emission from the thermal decomposition must be considered, too.

Gate location

Gate location is the most important part during gate design and has to be done at the beginning of the mold designing process. Changing the wrong gate position incurs much greater cost than other factors. The following factors have to be considered when finding a proper location.

1) Consider the requested tool type and runner system: A regular 2-plate mold save for a direct gate has its gate nearby the product and a 3-plate mold or hot runner has its gate in the product, itself.

2) Consider the gate position, mold type, and material: The disk gate needs a 2-plate cold runner or hot runner mold and a pin point gate needs a 3-plate mold.

3) Wall thickness variation: The gate has to be at the thickest part of the product. When the gate is placed at the thinner part, the thicker part has a packing control and causes shrinkage, distortion, and void problems.

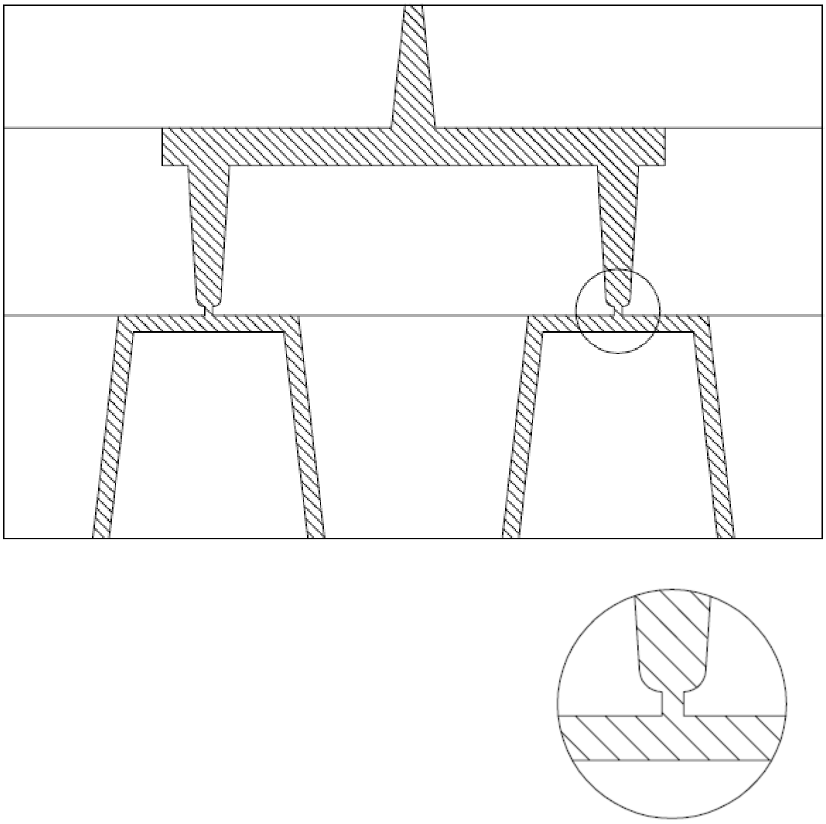
4) Weld line formation and location: Consider appearance problems and welded parts' strength drop.
5) Gate marks and residual stress at the gate part must be considered.
6) Consider flow pattern and shrinkage rate: flow-induced orientation provokes deformation and residual stress when it has uneven shrinkage.
   - A gate at the end tip of a long and narrow product structures the flow-induced orientation and by that minimizes residual stress.
   - A 3-dimensional symmetrical shape (e.g. cup, boxes) that has a radial flow should have its gate better installed at the center of the product.
   - A 2-dimensional-shaped (flat disk or square) product fits with gate in the center, but a square-shaped product needs to have its flow leader designed to minimize deformation. When flatness is important, a fan gate or multiple small gates have an advantage with linear flow.
7) Consider the running distance (runner and the cavity) by the number of gates and their location: this directly affects the injection pressure and runner's size, product thickness is indirectly affected.
8) Clamping force: when the gate placed at the clamping area is large, running distance gets longer and clamping pressure increases. In the opposite situation, shot imbalance occurs.
9) Core deflection: uneven shot nearby the core provokes core flection.
10) Venting: Need to clarify if the flow pattern from the designated gate position is venting at the parting line (or find out if it needs special venting). Also needs to be checked for gas traps.
11) Jetting: Occurs by the inertial force of rapid material inflow. In other words, while melted material inflows without bumping the wall or core of the mold, jetting occurs at the other corner of the cavity. To improve the jetting problem, avoid placing the gate at the open place and put the gate off-center of the parting line. In addition, design the gate targeting melted material's speed to be dramatically changed, so that frozen parts can not enter the cavity.

Deciding the gate shape and its size

The side gate is the most generous gate shape, in which the gate is placed at the parting line. Recommended side gate's thickness is 50~70% of the product's thickness and the width is 1.0~1.5 times of gate's thickness, and gate land's thickness is better to be 70~80% of gate's thickness or 0.8~1 mm.

A pin point gate can run an automatic injection molding and free from the gate location. Bigger products can achieve a balanced flow pattern by setting multiple gates and it even helps reducing the injection pressure so that it needs less clamping pressure. Smaller products can increase productivity by multi-cavities. However, it leaves gate marks at the surface and the runner gets longer so that scrap increases. Shear heat also increases at the gate and the runner.

Recommended pin point gate's gate diameter is 60~70% of the product's thickness and for gate land is 0.8~1.2 mm.

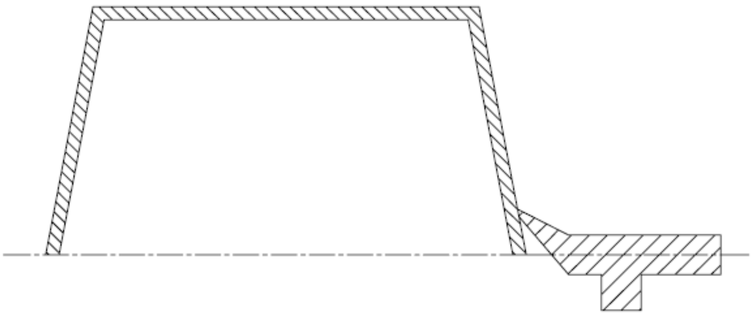


[Figure. 14] Example of pin point gate

A submarine gate is commonly used in multi-cavity molding for smaller products which have automatic molding at a 2-plate mold. The gate is placed at the sides and comparatively small so that it has an advantage on surface appearance, it needs special care of design due to complicated gate structure and defects from fragmentation of the gate during ejection.

A submarine gate's ideal diameter is 50% of the product's thickness, and gate land is preferable to be 0.5~0.8 mm. Undercut functions when the product is ejected, and the runner and gate have to be flexible enough, and it must not have any fragmentation when the gate and the product are detached. In other words, a smaller cross section area of the runner and gate extension is recommended for flexibility, and makes the point round where the runner and gate extension meet so that it does not break easily. When it is in a tapered shape, 30 ~ 45˚ are recommended for the center line and parting line.

The submarine gate is comparatively exquisite that it requires proper toughness rather than stiffness. Thus heat-treated (46~49Rc) pre-hardened H13 steel is recommended. As for, carburizing steel (SKD11), it breaks easily when hardness increases to 58~60Rc during the carburizing process.



[Figure. 15] Example of submarine gate

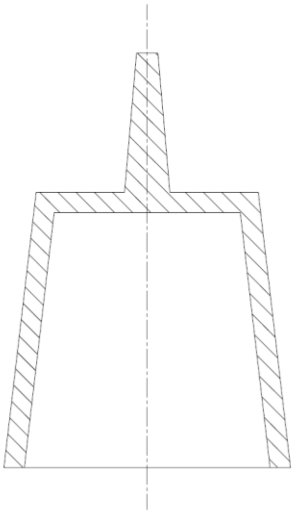
Gate freeze time

Gate freeze time affects not only the quality of the product but also the cycle time. Gate freeze time can be calculated by the formula of theoretic freezing time. By changing the transformation temperature to transition temperature in the theoretical freezing time formula, theoretical freezing time can be considered as a gate freezing time.

$$S = \frac{t^2}{\pi^2 \alpha} \ln\left[\frac{4}{\pi}\frac{(T_c - T_m)}{(T_x - T_m)}\right] \quad , \qquad \alpha = \frac{R}{C_p \rho}$$

S = theoretical freeze time
t = maximum thickness
a = material's thermal diffusion coefficient
R = material's heat conductivity
Cp = material's specific heat
Tx = material's ejection temperature
Tm = mold temperature
Tc = cylinder temperature

To choose the gate, consider the product's shape, number of cavities, performance, appearance, and economic feasibility. Location of gate is better to be close to the thick wall part because the existing pressure stays stable during the freezing process.



[Figure. 16] Example of direct / sprue gate

1) Direct (sprue) gate
Commonly used when forming only a single product or putting the gate at the bottom of the product manually. Even though the injection pressure is directly applied to the product can cause deformation from residual stress, mold structure itself is the simplest.

Classification	Contents
Explanation	- Sprue performs the function of the gate
Application	- Circle or symmetrically shaped products - Buckets, helmets, boxes, cups, disk-shaped parts
Mold	- 2-plates mold
Ejection type	- Manual
Advantages	- Effective pressure transmission - Ensuring better roundness for circle-shaped products - Easy molding and simple mold design
Disadvantages	- Longer freezing time delays the cycle time - Only 1 cavity can be molded - Leaves gate marks
Precaution	- To avoid appearance failure from the frozen material at the end of the nozzle, dimples are required to replace the cold slug. - To minimize the residual stress and orientation at the gate part, install ring shaped ribs. - After eliminating the gate, to flatten the floor, install recesses(凹) at the rim or the floor of the gate.

2) Side gate
Most commonly used especially for the multiple-cavities mold. Rectangular and half-round gates can be installed at the side of the product.

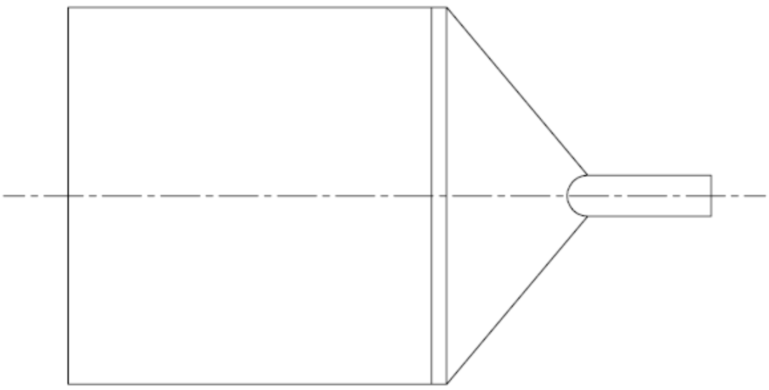
Classification	Contents
Explanation	- Most common gate form. (mostly using rectangular cross section) - Place the gate at the margin of the product
Application	- Applies to most plastic products - Product that needs to have a gate at the parting line
Mold	- 2-plates mold
Ejection type	- Manual
Advantages	- Easy mold processing - Easy to install multi-cavity

Disadvantages	- Gate marks at the side of the product - Automation is not available due to gate cut

3) Fan gate

Although similar to the side gate, larger width of the gate and its fan shape, comparatively larger product mainly uses it.

Classification	Contents
Explanation	- Similar to side gate but comes from a larger gate exit where the flow shear is larger and even
Application	- Thin products - Product with side gates but has possible jetting problems
Mold	- 2-plates mold
Ejection type	- Manual
Advantages	- Improved orientation at the gate part and minimized jetting problems - Reducing the residual stress and shear speed at the gate part
Disadvantages	- Requires extra equipment and personnel to cut the gate



[Figure. 17] Example of fan gate

4) Pin point gate

A pin point gate has a diameter of only 0.5~2.0 mm. Generally it does not need extra finishing. During the molding process, its gate freeze time is quick and has less deformation from the product's residual stress. Flow speed slows down when the cross section is small and flow marks occur easily nearby the gate.

Classification	Contents
Explanation	- Automatic molding available - Flow balance and lower injection pressure can be achieved by using multiple cavities (for larger products)

	- For smaller products, productivity can be improved by multi-cavities
Application	- A multiple smaller product molding - Larger products using multi-gate - Product that allows small gate marks at the exterior
Mold	- 3-layer mold
Ejection type	- Automatic
Advantages	- Smaller gate marks / easy gate separation - Flexible runner design (no limits on gate position) - Wide range of molding machines (clamping force can be reduced by using multiple gates)
Disadvantages	- High injection pressure and excessive shear heat at the gate part - Generates comparatively more scrap
Precautions	- Install dimples and recessed gates to minimize gate marks and improve the appearance

(4) Vent

To prevent gas carbonized marks, mold deposits, and short shot, gas vents have to be designed properly. Insufficient air vents can lead to short shot or carbonization can appear by the rapid temperature increase by the adiabatic compression of the air in the cavity. Recommended air vent depth at the cavity part is 0.01~0.02 mm and width of 2~6 mm. It is recommended to have a deeper air vent for the runner part which is 0.015~0.025 mm.

(5) Mold cooling

Space between the cooling lines must provide even temperature to each part of the product. Cooling line allocation must consider the physical restriction in the mold. Choosing between parallel or serial cooling lines must be considered carefully. It can affect the flow balance, refrigerant temperature change, and maintenance of the cooling system.

Mold cooling

Mold temperature control highly affects quality and productivity of the product. Quality- wise, wrong mold temperature control leads to appearance failure and deformation. Cooling line design has to achieve an even and uniform temperature.

There are 2 types of mold temperature control: coolant circulation type and heater type. The coolant circulation type circulates the refrigerant to exchange the heat with the mold. The heater type inserts the heating bar and directly heats up the mold.

The main factors to be considered for the cooling line design are its location, cooling method, and diameter. Diameter shall consider the flow speed. The cooling efficient depends on Reynolds number (Re) and the cooling line's internal diameter shall have 10,000~30,000 Reynolds number.

$$R_e = d \cdot \rho_L \cdot \upsilon / \mu$$

(d : pipe(cooling line)'s internal diameter, ρ_L : density of refrigerant, υ : kinematic (coefficient) of viscosity of the refrigerant, μ : viscosity of the refrigerant))

The smaller the internal diameter, the bigger the pressure loss from the flow path resistance and flow volume decreases, which brings lower cooling efficiency. Thus, flowing volume has to be increased as the internal diameter increases to improve cooling efficiency.

From the uniform cooling point of view, it is idealistic to have the same distance between the cooling circuit and the product, because of complicated shape and eject pin location it is hard to place it densely and is better to keep a certain distance. The mold and the material has 100~400 times of the thermal conductivity such that the thermal conductivity time is quick enough from the cavity part to the cooling line wall part. Thus, it can disturb the uniform cooling. To avoid it, the recommended distance between each cooling line is 5 times that of the internal diameter of the cooling line and 3 times between the mold surfaces and the cooling line.

(6) Undercut

There are 3 things to consider while designing the ejection plate and the ejection housing. First, the support bar arrangement must be considered to avoid clamp flexure from the moving plate. Secondly, bushing and guide pins are needed for the mold's array. Lastly, install the return pin for to protect the cavity from damage.

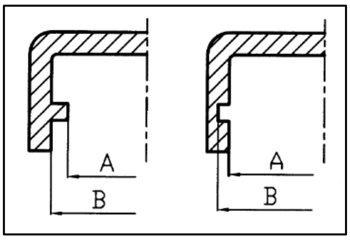
The ejection plate's forward/backward movement is powered by hydraulic force. In addition, the hydraulic cylinder and spring functions return the ejection plate. The ejection system must consider the undercut that can appear from product design.

Any protrusion or hole from the product at the moving plate cannot be placed as those parts hold the moving plate, and can't be opened are called 'undercut'. Undercut can be avoided by using a slide core, rolling core (when the product is a screw type), modified core, or ejection pin.

Undercut can be ejected by force within the allowance of the material's elastic limit. In the case of POM, the allowance range is 5% and can be calculated by the following formulas depending on location.

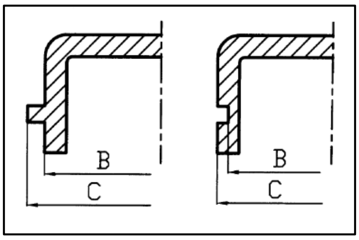
- Inner undercut:

Undercut allowance (%)= $\frac{B-A}{A} \times 100$



- Outer undercut:

Undercut allowance (%)= $\frac{C-B}{C} \times 100$



1) Slide core: Commonly used when the undercut is at the outside of the product. The undercut-shaped core moves parallel to the parting line by the angular pin. Cavity

design has to aim for the undercut shape when the mold is closed. During its opening, the core is separated from the undercut part along with the angular pin to help the product be ejected from the mold.

2) Modified ejecting pin: It is used when the undercut is small, multi-cavities are used, and limited space exists in the mold. Both inner and outer undercut can be used but is usually applied to the inner undercut.

3) Elastic core: Usually used with the smaller undercut and made by spring steel. Undercut ejects when the spring steel recovers through its elastic restoring force.

(7) Draft or taper

Draft, or taper, is an inclination at the vertical wall of the product that helps product to be easily ejected from the mold. Draft is designed in accordance with the product's shape, material, mold structure, finishing, embossing, and the quality level. Each material has a different recommendation and the range starts from 1/8˚ to several levels. It also considers ejecting and specific requirements for the product. Draft at the inner or outer side of the side wall's surface has to be symmetrical. The bigger the draft is, the less the ejection problem occurs.

The bigger the better the draft is, but the general recommendation is 1/2 ~ 1˚. This sometimes makes the core side's draft smaller in proportion to the cavity to keep the products in the core for better ejection in the case of thin products, but in most of cases, the cavity and the core's draft are the same.

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

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

KEP Europe GmbH
Rheingaustrasse 190-196 D-65203 Wiesbaden, Germany
Tel +49 (0)611 962-7381, Telefax +49 (0)611 962-9132

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