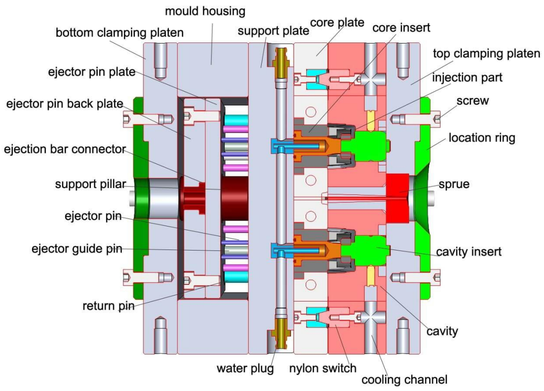
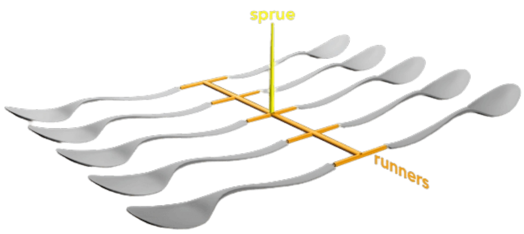


It is imperative to design the injection mold correctly to produce a functional plastic component. Part quality and productivity are directly linked to molding operation profitability, and mold design significantly impacts these factors. The following section provides general guidelines for designing thermoplastic part molds by [Topworks Plastic Mold](#).



Sprue Bushings

The injection molding machine's nozzle is connected to the mold's runner system by sprue bushings. Sprues should have a short length to reduce material consumption and cycle times as much as possible. The bushing should be smooth, tapered internally, and polished in the drawing direction (draw polished) to ensure proper separation of the sprue and the bushing. A positive sprue puller is also recommended. Figure 36 shows three standard sprue puller designs.

The shape of the runner

From the sprue to the gate, molten material is conveyed by runners. The runner's section must have the maximum cross-sectional area and the minimum perimeter.

To be a successful runner, athletes must have a high volume-to-surface-area ratio.

Creating such a section allows for less heat loss, molten resin solidification prematurely, and pressure drop in the runner system.

A circular profile is ideal for runners. In Figure 37, we can see an example of this type of runner. Full-round runners are the most efficient type, but they are also the most expensive to supply because they must be cut into both mold halves.

The trapezoid is a less expensive and equally effective section. Figure 37 shows the trapezoid runner with a depth equal to its base width and a taper of 2 to 5 degrees on each side. As a result, the surface area-to-volume ratio is good.

Due to their low volume-to-surface area ratio, half-round runners are not recommended. The figure below demonstrates why. It is clear from the comparison that the half-round runner's design has a poor perimeter-to-area ratio compared with the trapezoidal design if the inscribed circles are viewed as flow channels for the polymer.

Diameter of the runner

Ideally, many factors affect the size of the runner diameter - including the volume, the length of the runner, the gate size, and the cycle time. To avoid early freeze-off or excessive cycle times, runners should have diameters equal to or smaller than the maximum part thickness. Small enough to maintain an adequate cycle time but large enough to minimize pressure loss. Computer analysis has shown that smaller runner diameters have successfully been used to maintain melt temperature and enhance polymer flow because the smaller runner diameter increases material shear heat.

Running large runners is not economical due to the amount of energy required to form the material and then regrind it after it has solidified.

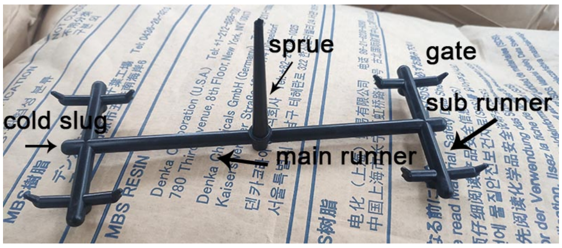
- **Diagram of a runner**

As shown in Figure 38, molds that contain multi-cavity parts should use an "H" runner system that is balanced. Balancing the runner system ensures the same rate and pressure filling in all mold cavities. Multi cavity molds and parts with similar geometries do not exist in all molds. The Dow Plastics Company offers computer-aided mold filling analysis as a service to its customers, allowing us to ensure that you will get a better-balanced fill for your part design. You can design molds with the following features when using mold filling simulation programs:

It runs at the proper temperature at minimum cost, reduces regrind, lowers barrel temperature and pressure, and saves energy while minimizing product degradation.

An artificially balanced runner system fills family tool cavities simultaneously and pressure, allowing cavities that are easier to fill to be fully stuffed.

- **Cold Slug Wells**



Runner intersections should be overrun by at least one diameter of the primary runner. This is shown in Figure 39. Melt traps or cold slug wells result from this process. This is because cold slug wells allow the molten mass to flow more readily into the cavity after catching the colder, higher-viscosity polymer at the front of the molten mass. So that cold material cannot adversely affect the final properties of the part, the cold slug well restricts the entry of cold material into the cavity.

- **Runnerless Molds**

By extending the melting chamber of the molding machine, runnerless molds extend the machine's nozzle and are different from traditional cold runner molds (Figure 40). This system maintains the plasticizing barrel's polymer melt at a temperature and viscosity equivalent to the polymer in the runnerless system. Runnerless molds can either be insulated or heated (heated runner).

- **Insulated Runners**

An insulated runner system (Figure 41) involves flowing molten polymer into the runner. The polymer forms a solid plastic layer on the runner walls during cooling, creating an insulating layer. By reducing the diameter of the runner and maintaining its temperature, the insulating layer reduces the melting portion of the melt as it waits for the next shot.

During each injection, the molten polymer in the runners should be injected into the mold while the run volume should not exceed the volume of the cavity. This full consumption is required to prevent excessive buildup of the insulation skin and minimize any drop in melt temperature.

With insulated runner systems, there are several advantages over conventional runner systems:

- Less sensitivity to the requirements for balanced runners.
- Reduction in material shear.
- More consistent volume of polymer per part.
- Faster molding cycles.
- Elimination of runner scrap – less regrind.
- Improved part finish.
- Decreased tool wear.

It does, however, have some disadvantages as well. Making and operating the mold requires a high level of technology, which leads to:

- Mold design is generally more complex.
- Mold costs generally increase.
- The start-up process is more complex until it runs correctly.
- Polymer melt may degrade thermally.
- Color changes are more challenging.
- Maintenance costs increase.

Hot Runners

In Figure 42, we see a hot runner system, which is the most widely used runnerless mold design. Mold design can be more flexible using this system - especially for large, multi-cavity molds - since it allows greater control over melt temperatures and other processing conditions.

Hot runner molds retain some of the advantages of insulated runners over conventional cold runners while eliminating some disadvantages. They require less time to start up. Hot runner molds have the following significant disadvantages over cold runner molds:

Design, manufacture, and operation of more complex molds.

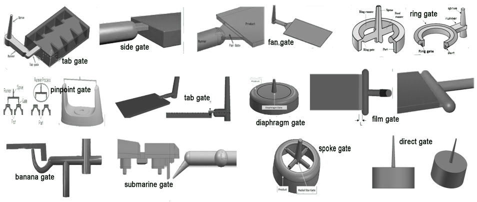
Costs are significantly higher.

In addition to these disadvantages, heated manifolds need to be installed, the heat provided by the manifold must be balanced, and polymer hang-ups must be minimized.

It maintains an entirely molten polymer from the machine nozzle to the mold gate by acting as an extension of the machine nozzle. A heating element and controls are provided on the manifold to maintain the desired melt temperature. The elements are challenging to install and control. Additionally, it is difficult to insulate the rest of the mold from the heat generated by the manifold, keeping the cavity cool in a cyclic manner.

Thermal expansion is another problem with mold components. Manifold alignment and cavity gate alignment are crucial aspects of mold design.

Today, molds without runners are available from a wide selection of suppliers. Such systems are usually chosen based on factors such as cost and design - be careful when picking one for a specific application.



● Gates

Gates are transition zones between the runner and the part, and their design should allow for easy filling of the mold.

● Gate Size

Runners and parts should be separated easily with gates that are small enough. Despite this, they must be large enough to prevent premature freezing-off of the polymer flow, which may adversely affect the consistency of the part dimensions. Specify the gate size as small as possible while still maintaining consistent filling of the mold. After you have a consistent mold filling, increase the gate size until it is consistent. For gates, we recommend a diameter of 0.75 mm, which should not exceed the diameter of the runner or sprue. The nominal wall thickness of gates is often half that of the part.

Place of the gate

Gate placement is critical to the performance of the finished part. Here are some guidelines to help you determine the best gate placement.

Aesthetics

A visible surface should not have remnants of a gate. Whenever possible, place gates away from visible surfaces.

Stress

Putting gates in high-stress areas is not a good idea. There are high residual stresses near the gate area because of both the gate itself and the degating of the part that may be needed. Stress is also concentrated by the rough surface left by the gate.

Pressure

Place gates in thick sections of parts to ensure adequate pressure for packing-out, thereby preventing sinks and voids.

Orientation

The location of the gate affects the molecule's orientation. When the flow channel depth decreases in thin sections, molecular orientation becomes more pronounced. Molecular alignment is mainly determined by flow stress orientation.

Uniaxial strength is a result of high degrees of orientation. Such parts are generally resistant to only one direction of forces. Whenever molten polymer enters a cavity, place a gate so that an obstruction diverts the flow. This will minimize molecule orientation.

Weld Lines

Generally, injection molding should equalize the flow length throughout the cavity by using gates. You should also minimize the number and length of welding lines by placing gates.

The figure 43 illustrates how weld lines are formed and how they can be avoided. If the gate has to be placed next to the weld line, ensure it is as close as possible to the obstruction to maintain a high melt temperature and ensure a strong weld.

Filling

Choose gate locations where the polymer will impinge against walls (or other projections such as pins). As a result, the part will not have any jetting and will also be free of flow marks and gate-blush.

Vents

plastic	Depth mm	plastic	Depth mm
PE	0.02	PA(GF)	0.03
PP	0.02	PA	0.02
PS	0.02	PC(GF)	0.06
ABS	0.03	PC	0.04
SAN	0.03	PBT(GF)	0.03
ASA	0.03	PBT	0.02
POM	0.02	PMMA	0.04

Ventilation is needed in all mold cavities to let the air out when the polymer flows into them. Short shots, weak weld lines, burn marks, or high packing pressures can be caused by inadequate venting.

The economics of mold construction often limit the number of vents in a mold. The location of vents should be specified on part prints.

High melt flow materials, in general, require smaller vents than low melt flow materials.

In polycarbonate, a flow rate of 3 melts may be sufficient to mold with a flow rate of

No vestige of the vent can be seen in this 0.08 mm (0.003") vent. However, the part may appear tattered at the vent entrance when a polycarbonate that melts at 22 is used in the same mold.

Dimensions of the vent

At least 0.25 mm from the edge of the mold cavity, the vent depth should be determined as indicated in Table 16, between 0.02 mm and 0.05 mm. As the vent depth increases, the vent width should be a minimum of 3mm, and the vent depth should be a minimum of 0.75mm.

Gates and vents should have the same size. Once the molding is satisfactory, start with shallow vents and enlarge them if necessary. Smaller mold vents tend to become clogged, limiting or eliminating the airflow out of the mold cavity. Parts located at vent locations can flash if they have large openings.

Vent Location

Parting lines of molds can be equipped with vents located anywhere along their length, particularly at last-to-fill points. In general, 25 mm is a reasonable pitch for vents. By grinding flat spots on ejector pins along the major axis of a cavity or rib, blind ribs or bosses can be incorporated into the mold.

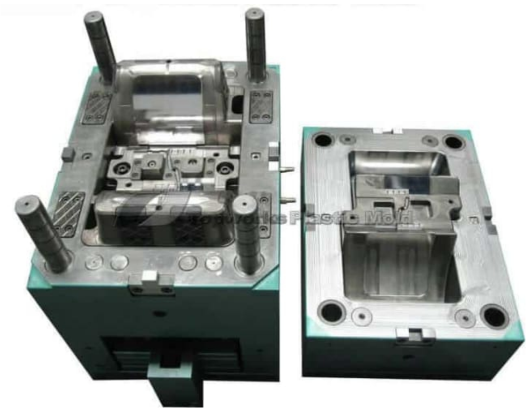
Alternatively, injection mold can use sintered metal inserts for venting. The inserts let gas pass through them but prevent the polymer from clogging them. Sintering metal inserts should be applied only to non-visible surfaces as a last resort.

Mechanisms of ejection

When designing plastic parts, part ejection from the mold must be considered in the concept phase. Developing components with the ejection process in mind greatly reduces the need for expensive and complicated ejection systems later on, as a part becomes difficult to eject.

- Designing the ejection mechanism should take into account four factors:
- The geometry and shape of the part.
- Material and thickness of the wall.
- The volume of production to be produced.
- The position of the component concerning the parting line.

The designer will usually have a better idea of which mechanism to use based on these factors. Following are some guidelines that will help you select a mechanism.



● Ejector or Knockout Pins

The ejection methods outlined here are extremely common and inexpensive.

Ideally, pins should be located at the edges of corners, ribs, bosses, and other features that make ejection more difficult. You can choose from pin geometries such as stepped pins, blade pins, and valve pins.

● Other Methods

The sleeves of ejectors are commonly used around the bosses of parts. Plates or rings are commonly used in thin-walled containers mold. In air ejection, the air is contained in part long enough to blow the part off the mold (a flat part would be impossible).

For the part to be ejected effectively, the designer must calculate the surface area regardless of the ejection method selected. Ejection mechanisms can damage the surface of a part of their surface area is insufficient.

To calculate the force necessary to remove a part from a mold, you can use the following equation.

● Cooling

For just-molded plastic parts to be ejected from molds as soon as possible, molds must be cooled to remove heat. A heat-transfer fluid is circulated through passages drilled or machined in a mold to cool it. The mold surfaces, the core, and each cavity should also include direct cooling passages and passages for cooling in the molding block or plates. The cooling of the just-molded article must be efficient and effective to remove the heat and allow its ejection. Since cooling accounts for 70 to 80% of cycle time, inefficient cooling can be very costly.

The diameter of cooling channel bores should range from six to ten millimeters. Smaller pipes should not be used except in extreme cases. Similarly, sized hoses are required to connect passages in the mold.

A turbulent flow of the cooling fluid - either water or ethylene glycol/water - maximizes the cooling rate. Heat is transferred three to five times more efficiently with the turbulent flow than with non-turbulent flow.

The material used to make the mold also impacts the cooling rate. Copper molds transfer heat twice as effectively as carbon steel molds and four times as effectively as stainless steel molds. The use of beryllium copper does not automatically translate to four times the speed of stainless steel mold cycles. In contrast, thin-wall parts can run more quickly in beryllium copper molds.

Molds made of beryllium copper should not be used for thermoplastics requiring high mold temperatures. The high thermal conductivity of beryllium copper makes it difficult to maintain a sufficient level of heat economically due to heat loss to the environment. Dow Plastics offers the computer-assisted analysis of mold-cooling networks to ensure that your molded part is adequately and uniformly cooled.