Chapter 5

Product and Mold Design

Product Design
Mold Design

Product Design

Although component design in thermoplastics is complex, following a few fundamental principles will help you minimize problems during molding and in part performance. Of course, the guidelines given here are general. Depending on the particular requirements of the part, it may not always be possible to follow all of our suggestions. But these guidelines, in furthering your understanding of the behavior of thermoplastics, can help you effectively resolve some of the more common design problems.

Nominal Wall Thickness

For parts made from most thermoplastics, nominal wall thickness should not exceed 4.0 mm. Walls thicker than 4.0 mm will result in increased cycle times (due to the longer time required for cooling), will increase the likelihood of voids and significantly decrease the physical properties of the part. If a design requires wall thicknesses greater than the suggested limit of 4.0 mm, structural foam resins should be considered, even though additional processing technology would be required.

In general, a uniform wall thickness should be maintained throughout the part. If variations are necessary, avoid abrupt changes in thickness by the use of transition zones, as shown in Figure 25. Transition zones will eliminate stress concentrations that can significantly reduce the impact strength of the part. Also, transition zones reduce the occurrence of sinks, voids, and warping in the molded parts.

A wall thickness variation of ±25% is acceptable in a part made with a thermoplastic having a shrinkage rate of less than 0.01 mm/mm. If the shrinkage rate exceeds 0.01 mm/mm, then a thickness variation of ±15% is permissible.

Radii

It is best not to design parts with sharp corners. Sharp corners act as notches, which concentrate stress and reduce the part's impact strength. A corner radius, as shown in Figure 26, will increase the strength of the corner and improve mold filling. The radius should be in the range of 25% to 75% of wall thickness; 50% is suggested. Figure 27 shows stress concentration as a function of the ratio of corner radius to wall thickness, R/T.

Draft Angle

So that parts can be easily ejected from the mold, walls should be designed with a slight draft angle, as shown in Figure 28. A draft angle of ½° draft per side is the extreme minimum to provide satisfactory results. 1° draft per side is considered standard practice. The smaller draft angles cause problems in removing completed parts from the mold. However, any draft is better than no draft at all.

Parts with a molded-in deep texture, such as leather-graining, as part of their design require additional draft. Generally, an additional 1° of draft should be provided for every 0.025 mm depth of texture.

Figure 25 – Suggested Design for Wall Thickness Transition Zone

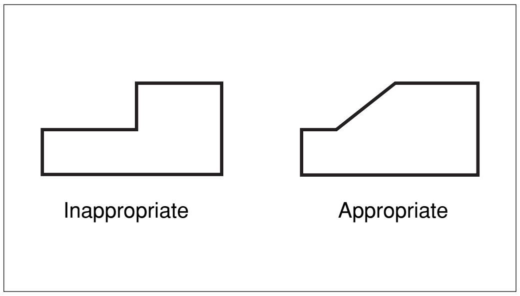


Figure 26 – Suggested Design for Corner Radius

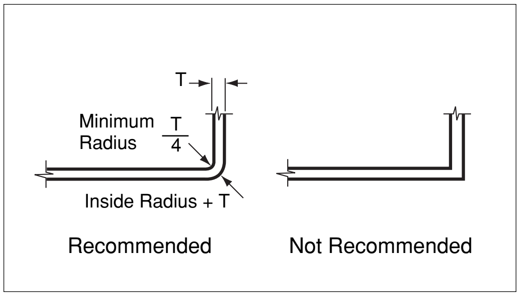


Figure 27 – Stress Concentration as a Function of Wall Thickness and Corner Radius

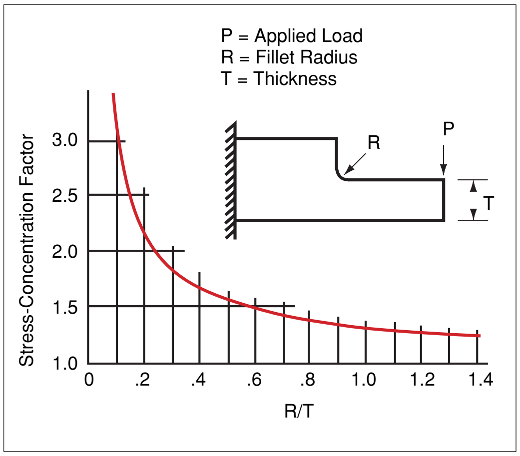
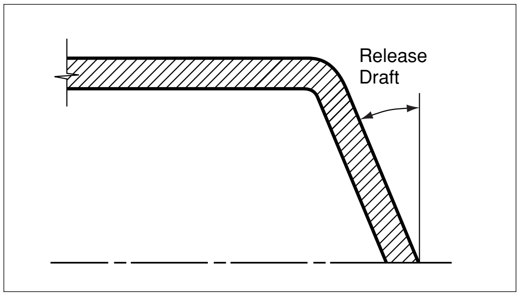


Figure 28 – Exaggerated Draft Angle



Ribs and Gussets

When designing ribs and gussets, it is important to follow the proportional thickness guidelines shown in Figures 29 and 30. If the rib or gusset is too thick in relationship to the part wall, sinks, voids, warpage, weld lines (all resulting in high amounts of molded-in stress), longer cycle times can be expected.

The location of ribs and gussets also can affect mold design for the part. Keep gate location in mind when designing ribs or gussets. For more information on gate location, see page 66. Ribs well-positioned in the line of flow, as well as gussets, can improve part filling by acting as internal runners. Poorly placed or ill-designed ribs and gussets can cause poor filling of the mold and can result in burn marks on the finished part. These problems generally occur in isolated ribs or gussets where entrapment of air becomes a venting problem.

Note: It is further recommended that the rib thickness at the intersection of the nominal wall not exceed one-half of the nominal wall in HIGHLY COSMETIC areas. For example, in Figure 29, the dimension of the rib at the intersection of the nominal wall should not exceed one-half of the nominal wall.

Experience shows that violation of this rule significantly increases the risk of rib read-through (localized gloss gradient difference).

Figure 29 – Example of Rib Design

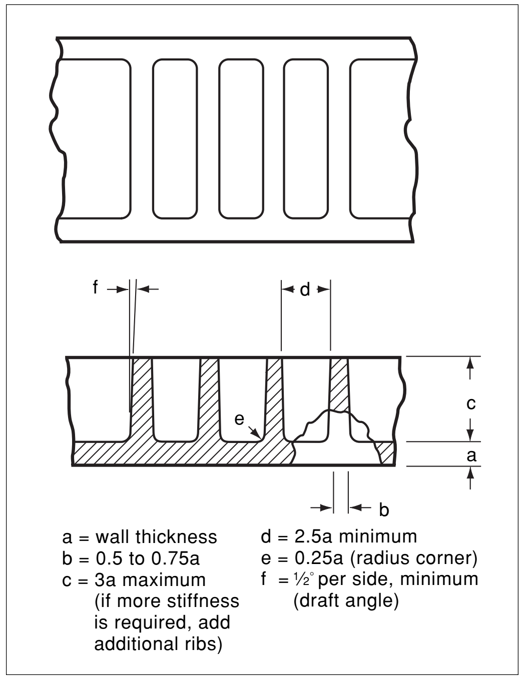


Figure 30 – Example of Gusset Design

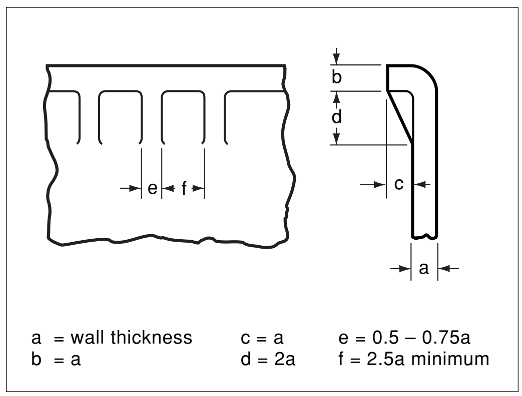


Figure 31 – Recommended Design of a Boss Near a Wall (with Ribs and Gussets)

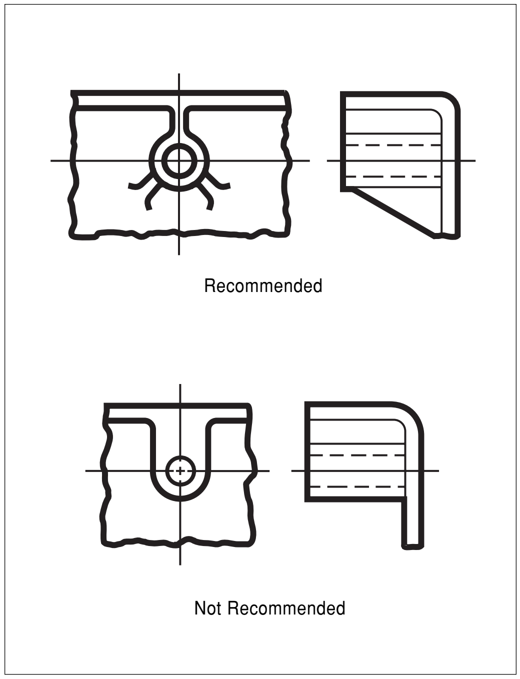
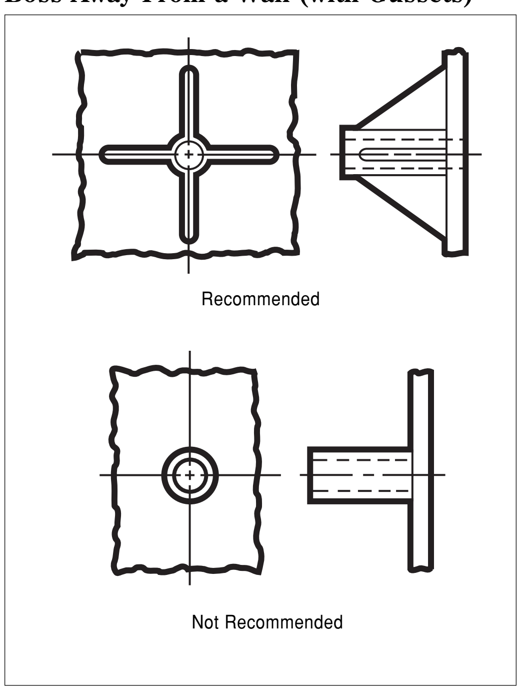


Figure 32 – Recommended Design of a Boss Away From a Wall (with Gussets)



Bosses

Bosses are used in parts that will be assembled with inserts, self-tapping screws, drive pins, expansion inserts, cut threads, and plug or force-fits. Avoid stand-alone bosses whenever possible. Instead, connect the boss to a wall or rib, with a connecting rib as shown in Figure 31. If the boss is so far away from a wall that a connecting rib is impractical, design the boss with gussets as shown in Figure 32.

Figures 33 and 34 give the recommended dimensional proportions for designing bosses at or away from a wall. Note that these bosses are cored all the way to the bottom of the boss.

Figure 33 – Recommended Dimensions for a Boss Near a Wall (with Rib and Gussets)

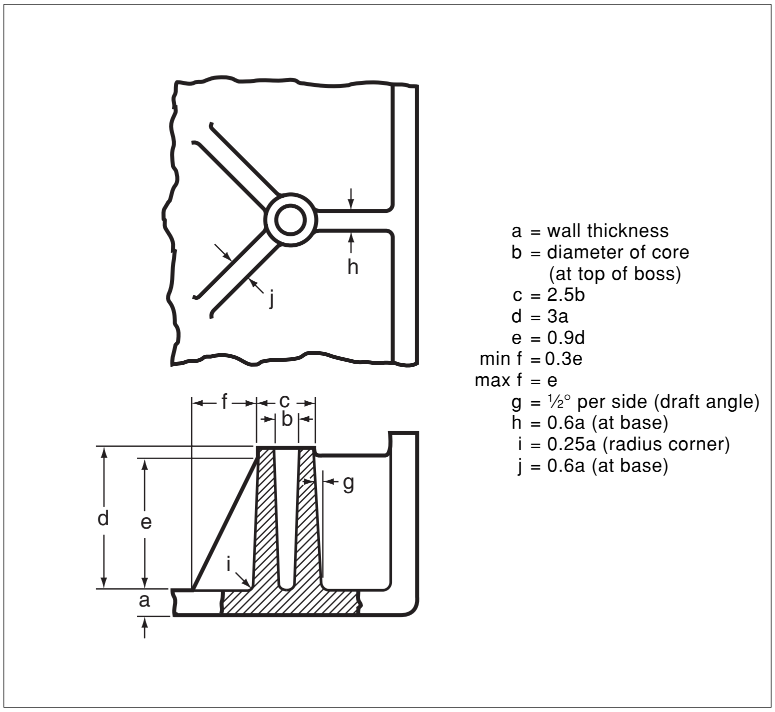
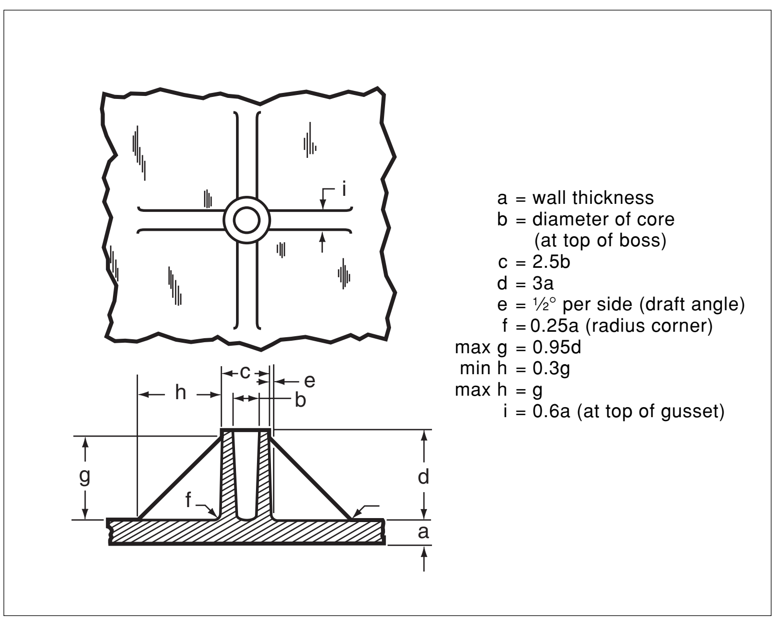


Figure 34 – Recommended Dimensions for a Boss Away From a Wall (with Gusset)



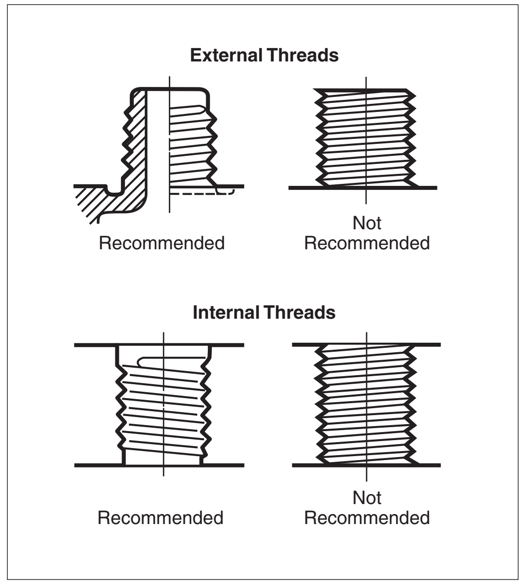
Threads

Molded-in threads can be designed into parts made of engineering thermoplastic resins. Threads always should have radiused roots and should not have feather edges – to avoid stress concentrations. Figure 35 shows examples of good design for molded-in external and internal threads. For additional information on molded-in threads, see page 105. Threads also form undercuts and should be treated as such when the part is being removed from the mold i.e., by provision of unscrewing mechanisms, collapsible cores, etc. Every effort should be made to locate external threads on the parting line of the mold where economics and mold reliability are most favorable.

Undercuts

Because of the rigidity of most engineering thermoplastic resins, undercuts in a part are not recommended. However, should a design require an undercut, make certain the undercut will be relieved by a cam, core puller, or some other device when the mold is opened.

Figure 35 – Recommended Design for Molded-in Threads



Mold Design

Proper design of the injection mold is crucial to producing a functional plastic component. Mold design has great impact on productivity and part quality, directly affecting the profitability of the molding operation. This section provides general guidelines for the design of a good, efficient mold for making thermoplastic parts.

Figure 36 – Three Common Sprue Pullers

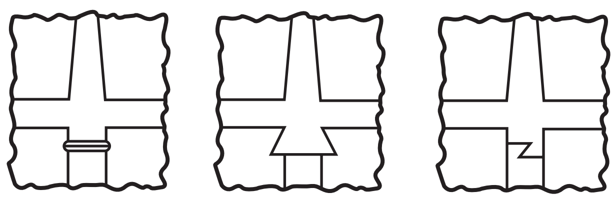
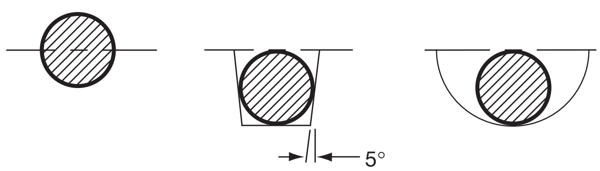


Figure 37 – Three Conventional Runner Profiles



Sprue Bushings

Sprue bushings connect the nozzle of the injection molding machine to the runner system of the mold. Ideally, the sprue should be as short as possible to minimize material usage and cycle time. To ensure clean separation of the sprue and the bushing, the bushing should have a smooth, tapered internal finish that has been polished in the direction of draw (draw polished.) Also, the use of a positive sprue puller is recommended. Figure 36 shows three common sprue puller designs.

Runner Geometry of Conventional Mold

Runner systems convey the molten material from the sprue to the gate. The section of the runner should have maximal cross-sectional area and minimal perimeter. Runners should have a high volume-to-surface area ratio. Such a section will minimize heat loss, premature solidification of the molten resin in the runner system, and pressure drop.

The ideal cross-sectional profile for a runner is circular. This is known as a full-round runner, as shown in Figure 37. While the full-round runner is the most efficient type, it also is more expensive to provide, because the runner must be cut into both halves of the mold.

A less expensive yet adequately efficient section is the trapezoid. The trapezoidal runner should be designed with a taper of 2 to 5° per side, with the depth of the trapezoid equal to its base width, as shown in Figure 37. This configuration ensures a good volume-to-surface area ratio.

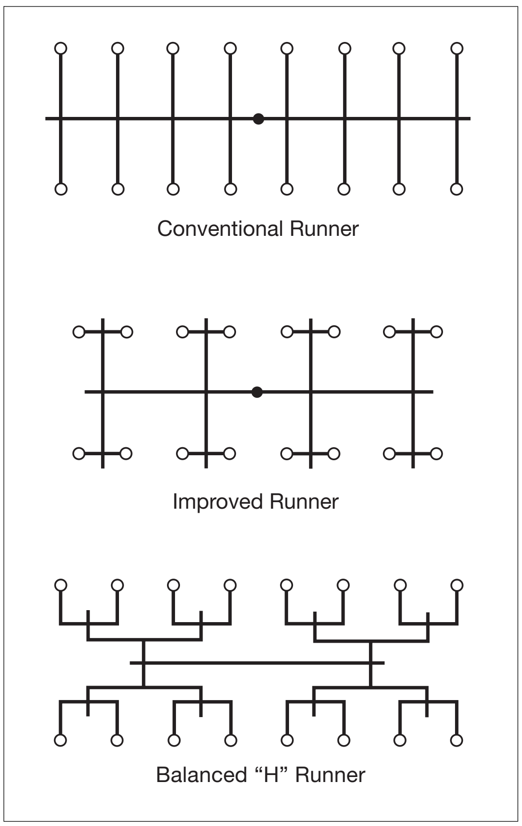
Half-round runners are not recommended because of their low volume-to-surface area ratio. Figure 37 illustrates the problem. If the inscribed circles are imagined to be the flow channels of the polymer through the runners, the poor perimeter-to-area ratio of the half-round runner design is apparent in comparison to the trapezoidal design.

Runner Diameter Size

Ideally, the size of the runner diameter will take many factors into account – part volume, part flow length, runner length, machine capacity gate size, and cycle time. Generally, runners should have diameters equal to the maximum part thickness, but within the 4 mm to 10 mm diameter range to avoid early freeze-off or excessive cycle time. The runner should be large enough to minimize pressure loss, yet small enough to maintain satisfactory cycle time. Smaller runner diameters have been successfully used as a result of computer flow analysis where the smaller runner diameter increases material shear heat, thereby assisting in maintaining melt temperature and enhancing the polymer flow.

Large runners are not economical because of the amount of energy that goes into forming, and then regrinding the material that solidifies within them.

Figure 38 – Runner System Layouts



Runner Layout

Similar multicavity part molds should use a balanced “H” runner system, as shown in Figure 38. Balancing the runner system ensures that all mold cavities fill at the same rate and pressure. Of course, not all molds are multicavity, nor do they all have similar part geometry. As a service to customers, Dow Plastics offers computer-aided mold filling analysis to ensure better-balanced filling of whatever mold your part design requires. Utilizing mold filling simulation programs enables you to design molds with:

- Minimum size runners that deliver melt at the proper temperature, reduce regrind, reduce barrel temperature and pressure, and save energy while minimizing the possibility of material degradation.
- Artificially balanced runner systems that fill family tool cavities at the same time and pressure, eliminating overpacking of more easily filled cavities.

Cold Slug Wells

At all runner intersections, the primary runner should overrun the secondary runner by a minimum distance equal to one diameter, as shown in Figure 39. This produces a feature known as a melt trap or cold slug well. Cold slug wells improve the flow of the polymer by catching the colder, higher-viscosity polymer moving at the forefront of the molten mass and allowing the following, hot, lower-viscosity polymer to flow more readily into the mold-cavity. The cold slug well thus prevents a mass of cold material from entering the cavity and adversely affecting the final properties of the finished part.

Figure 39 – Recommended Design of a Cold Slug Well

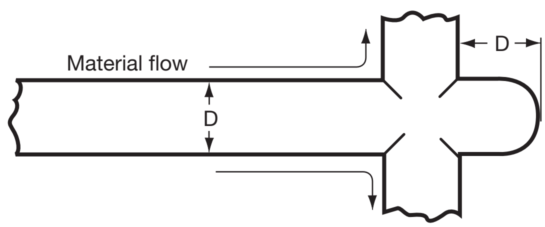
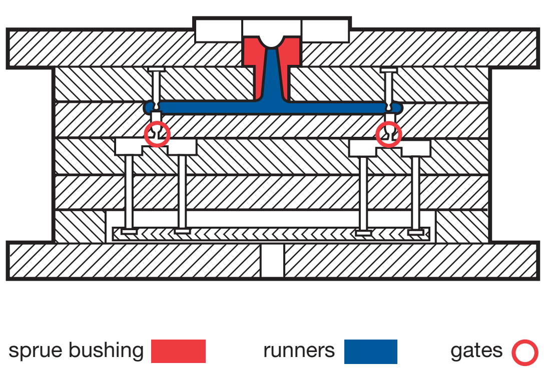


Figure 40 – Conventional Cold Runner Mold



Runnerless Molds

Runnerless molds differ from the conventional cold runner mold (Figure 40) by extending the molding machine's melt chamber and acting as an extension of the machine nozzle. A runnerless system maintains all, or a portion, of the polymer melt at approximately the same temperature and viscosity as the polymer in the plasticating barrel. There are two general types of runnerless molds: the insulated system, and the hot (heated) runner system.

Insulated Runners

The insulated runner system (Figure 41) allows the molten polymer to flow into the runner, and then cool to form an insulating layer of solid plastic along the walls of the runner. The insulating layer reduces the diameter of the runner and helps maintain the temperature of the molten portion of the melt as it awaits the next shot.

The insulated runner system should be designed so that, while the runner volume does not exceed the cavity volume, all of the molten polymer in the runners is injected into the mold during each shot. This full consumption is necessary to prevent excess build-up of the insulating skin and to minimize any drop in melt temperature.

The many advantages of insulated runner systems, compared with conventional runner systems, include:

- 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.

However, the insulated runner system also has disadvantages. The increased level of technology required to manufacture and operate the mold results in:

- Generally more complicated mold design.
- Generally higher mold costs.
- More difficult start-up procedures until running correctly.
- Possible thermal degradation of the polymer melt.
- More difficult color changes.
- Higher maintenance costs.

Hot Runners

The more commonly used runnerless mold design is the hot runner system, shown in Figure 42. This system allows greater control over melt temperatures and other processing conditions, as well as a greater freedom in mold design – especially for large, multicavity molds.

Hot runner molds retain the advantages of the insulated runner over the conventional cold runner, and eliminate some of the disadvantages. For example, start-up procedures are not as difficult. The major disadvantages of a hot runner mold, compared with a cold runner mold, are:

- More complex mold design, manufacture, and operation.
- Substantially higher costs.

These disadvantages stem from the need to install a heated manifold, balance the heat provided by the manifold, and minimize polymer hang-ups.

The heated manifold acts as an extension of the machine nozzle by maintaining a totally molten polymer from the nozzle to the mold gate. To accomplish this, the manifold is equipped with heating elements and controls for keeping the melt at the desired temperature. Installing and controlling the heating elements is difficult. It is also difficult to insulate the rest of the mold from the heat of the manifold so the required cyclic cooling of the cavity is not affected.

Another concern is the thermal expansion of the mold components. This is a significant detail of mold design, requiring attention to ensure the maintenance of proper alignment between the manifold and the cavity gates. (For more information on thermal expansion, see the section on thermal stress analysis, page 32.)

Currently there are many suppliers and many available types of runnerless mold systems. In most cases, selection of such a system is based primarily on cost and design limitations – be careful in evaluating and selecting a system for a particular application.

Figure 41 – Insulated Runner Mold

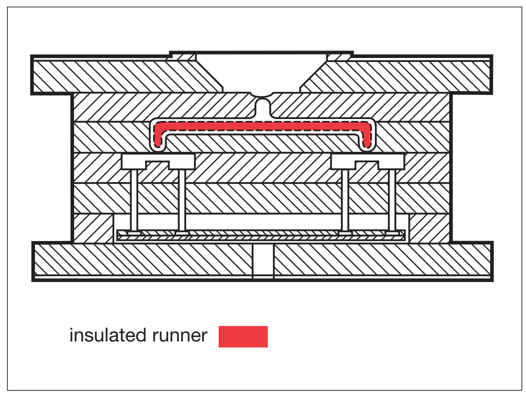
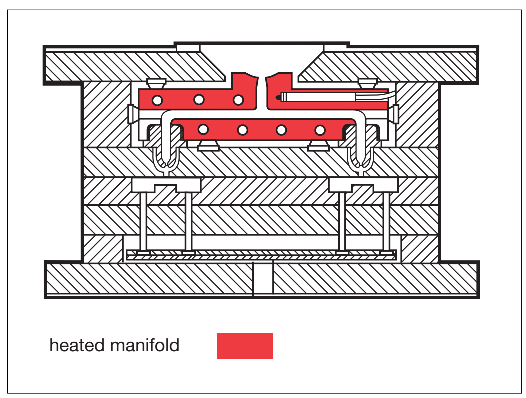


Figure 42 – Hot Runner Mold



Gates

The gate serves as a transition zone between the runner and the part, and should be designed to permit easy filling of the mold.

Gate Size

Gates should be small enough to ensure easy separation of the runner and the part. However, they should be large enough to prevent premature freezing-off of the polymer flow, which can affect the consistency of part dimensions. When specifying gate size, it is best to be “steel safe.” Start with a gate size smaller than you think will do the job, and increase the size until proper filling of the mold is achieved consistently. The minimum size we suggest for gate diameter is 0.75 mm, and, as a rule, it should not exceed the runner or sprue diameter. Gates are often designed to be half the nominal wall thickness of the part.

Gate Location

Correct location of gates has a critical effect on finished part performance. You should consider the following guidelines when determining gate location.

Appearance

Residual vestiges of a gate are normally unacceptable on a visible surface. Therefore, position gates on a non-visible surface whenever possible.

Stress

Do not place gates near highly stressed areas. The gate itself, and degating of the part that may be required, result in high residual stresses near the gate area. Also, the rough surface left by the gate creates stress concentrators.

Pressure

Place gates in the thickest section of a part to ensure ample pressure for packing-out the thick section, and prevention of sinks and voids.

Orientation

Gate location affects the molecular orientation of the polymer. Molecular orientation becomes more pronounced as the depth of the flow channel decreases in thin part sections. Because of flow stress orientation, most of the molecules align in the same direction.

High degrees of orientation result in parts having uniaxial strength. And such parts are primarily resistant only to forces acting in one direction. To minimize molecular orientation, position gates so that as soon as the molten polymer enters the cavity, the flow is diverted by an obstruction.

Weld Lines

In general, place gates to equalize flow length throughout the cavity. Also, place gates to minimize the number and length of weld lines.

Figure 43 shows how weld lines are formed and how they can be prevented. When weld lines are unavoidable, place the gate close to the obstruction forming the weld line – to maintain a high melt temperature and ensure a strong weld.

Filling

Select gate locations so that the polymer impinges against walls (or other projections, such as pins) as shown in Figure 44. This will eliminate jetting and also will help to prevent flow marks and gate-blush on the surfaces of the part.

Figure 43 – Positioning Gates to Eliminate Weld Lines

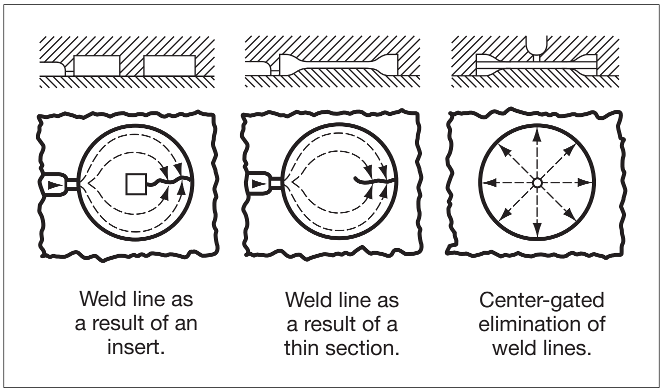
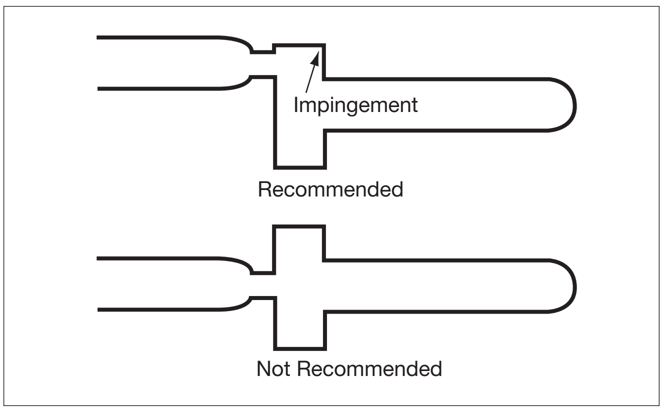


Figure 44 – Positioning Gates to Improve Polymer Flow



Types of Gates

Selecting the best type of gate for a given mold design is as important as the location and size of the gate. Many gate designs are readily available. The most commonly used gates are described here to help you to select the type best-suited for specific kinds of applications. See Figures 45 to 54.

Figure 45 – The sprue gate is recommended for single-cavity molds or for molds for circular parts requiring symmetrical filling. This gate is suitable for thick sections.

Figure 46 – The side, or edge gate is used for multicavity two-plate molds and is suitable for medium and thick sections.

Figure 47 – The pin gate (a three-plate tool) is often substituted for an edge gate to minimize finishing and provide a centrally located gate. It is good for applications that require automatic degating, but is suitable only for thin sections.

Figure 48 – The restricted, or edge pin gate allows simple finishing and degating. Like the pin gate (Figure 47), it is used only for thin sections.

Figure 49 – The tab gate is a restricted gate that prevents "jetting" and minimizes molding strain.

Figure 50 – The diaphragm gate is used for single-cavity molds for single-shaped parts that have a small or medium internal diameter.

Figure 51 – The internal ring gate is similar to the diaphragm gate, and is used for single-cavity molds to make ring-shaped parts having large internal diameters.

Figure 52 – The external ring gate is used for multicavity molds for ring-shaped parts when the diaphragm gate is not practical.

Figure 53 – The flash gate is a development of the edge pin gate for larger volume cavities.

Figure 54 – The geometry of the submarine gate.

Figure 45 – Sprue Gate

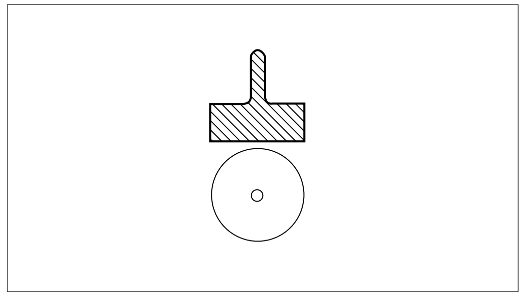


Figure 46 – Edge Gate

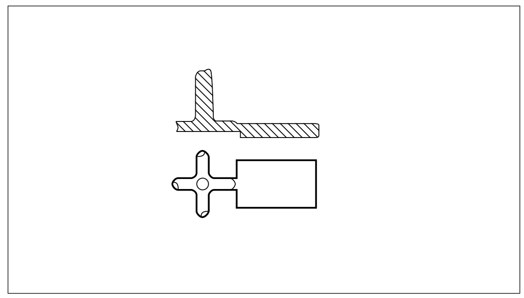


Figure 47 – Pin or Drop Gate (3-Plate Mold)

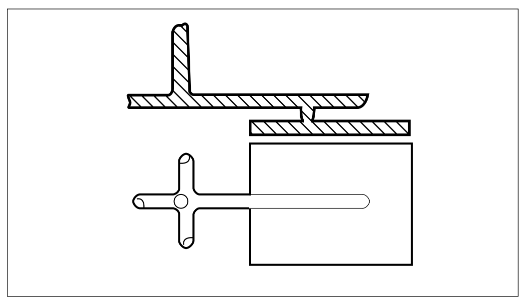


Figure 48 – Restricted Edge Pin Gate

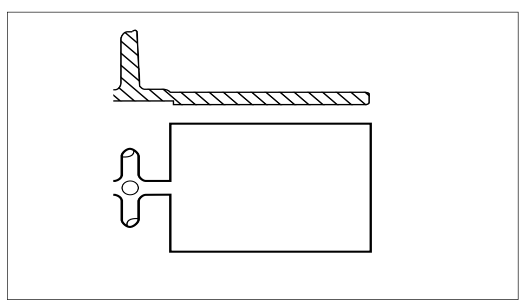


Figure 49 – Tab Gate

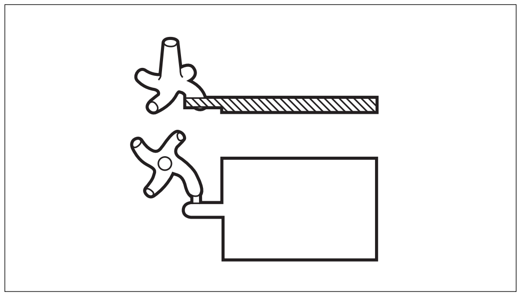


Figure 50 – Diaphragm Gate

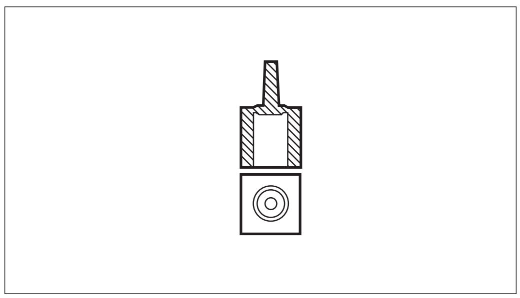


Figure 51 – Internal Ring Gate

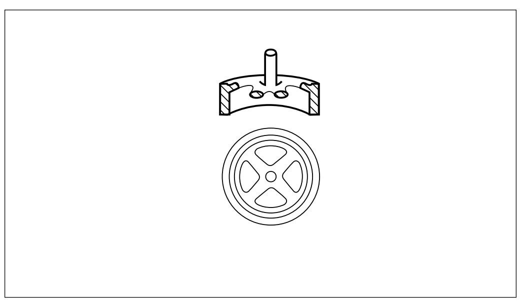


Figure 52 – External Ring Gate

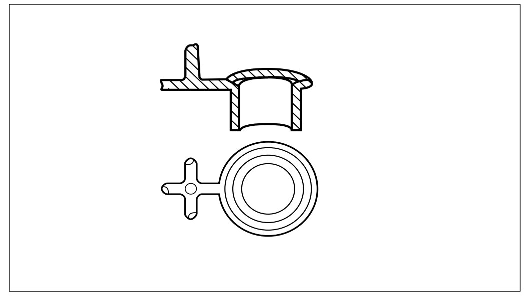


Figure 53 – Flash Gate

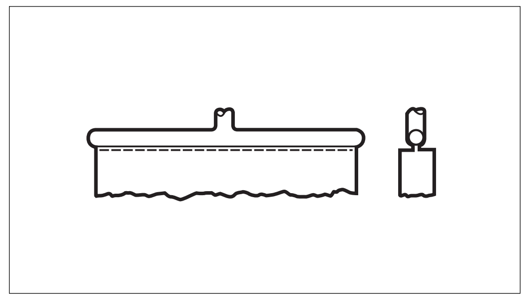
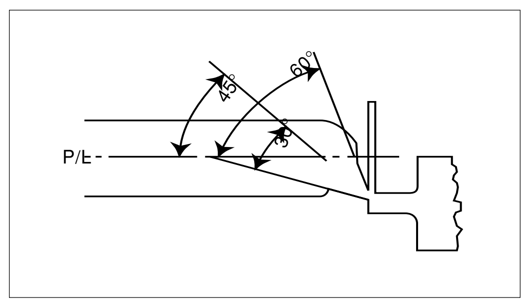


Figure 54 – Geometry of Submarine Gate



Vents

All mold cavities must be vented in order to release the air that is displaced when the polymer flows into them. Poor venting can result in short shots, weak weld lines, burn marks, and high molded-in stresses resulting from high packing pressures.

The number of vents in a mold is often limited by the economics of mold construction.

Good part design practices include specifying vent location on part prints.

Note: In general, higher melt flow materials must use smaller vents than a low melt flow version of the same material.

Example: Polycarbonate with a 3 melt flow rate may prove to mold sufficiently with a 0.08 mm (0.003") vent, showing no vent vestige. However, when a polycarbonate with a melt flow rate of 22 is run in the same mold, small vestiges may appear on the part at the vent entrance.

Vent Size

The vent depth should be as indicated in Table 16, from 0.02 mm to 0.05 mm for at least the first 0.25 mm distance from the edge of the mold cavity. The vent depth then should increase to a minimum of 0.75 mm to the outer edge of the mold and the vent width should be a minimum of 3 mm.

As in the sizing of gates, vents should be cut "steel safe." Begin with shallow vents and cut them larger, if needed, until molding is satisfactory. Vents that are too small tend to become clogged, reducing or eliminating their ability to release air from the cavity of the mold. Large vents can lead to flash on the part at the vent location.

Table 16 – Venting Techniques

Polymer	Depth of Vent mm (inch)
Polystyrene	.02-.05 (0.001-0.002)
ABS	.04-.06 (0.0015-0.0025)
PC/ABS	.02-.05 (0.001-0.002)
Polycarbonate	.02-.05 (0.001-0.002)
Polyethylene	.01-.02 (0.0005-0.001)

Figure 55 – Venting Geometry

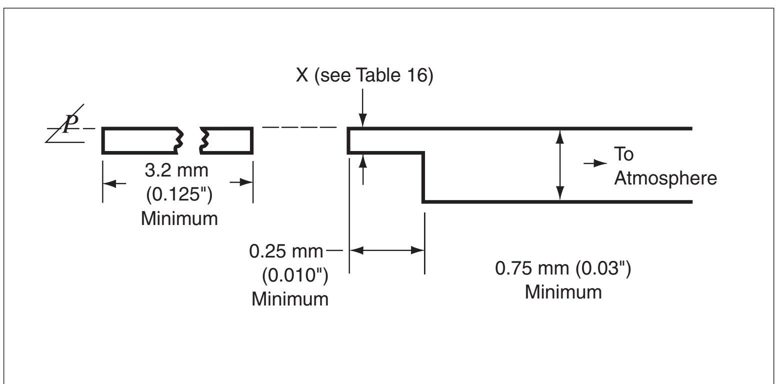
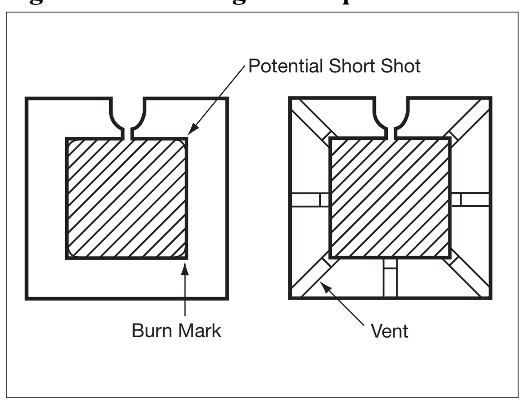


Figure 56 – Venting Techniques



Vent Location

Vents can be positioned anywhere along the parting line of the mold, particularly at last-to-fill locations as shown in Figure 56. A reasonable guide is to have vents spaced at 25 mm pitch. For blind ribs and bosses, vents may be incorporated into the mold by grinding flat spots along the major axis of an ejector pin or cavity.

Another option for venting is the use of sintered metal inserts. These inserts enable gas to pass into them but do not allow the polymer to clog them. Sintered metal inserts should be used only on non-visual surfaces and only as a last resort.

Ejection Mechanisms

When designing plastic parts, the method of part ejection from the mold must be considered in the concept phase. Designing with ejection in mind largely eliminates use of costly and complex ejection systems pressed into service later, when a part is difficult to eject.

Four factors should be considered in designing the ejection mechanism:

- Shape and geometry of the part.
- Type of material and wall thickness.
- Projected production volume.
- Component position relative to the parting line.

These factors will usually indicate to the designer which mechanism is most suited for the designed part. The following guidelines will help you decide on particular mechanisms.

Ejector or Knockout Pins

These are very common and inexpensive ejection methods. The pins are preferably located where changes in shape occur (at corners, ribs, bosses, etc.), because these features increase the difficulty of ejection. Among the various pin geometries are stepped pins, blade pins, valve pins, and standard flat pins.

Other Methods

Ejector sleeves are often used around part bosses. Stripper rings/plates are used with thin-wall containers. Air ejection is used to eject parts having an “enclosed” geometry (a flat part would not contain the air long enough to blow the part off the mold).

Regardless of the ejection method selected, the designer must calculate the area of part surface required if the part is to be ejected effectively. If the surface area of ejection is inadequate, the part surface can be damaged by the ejection mechanism. You can use the following equation to calculate the ejection force required to remove the part from the mold.

$$P = \frac{S_t \times E \times A \times \mu}{d\,[d/2t - (d/4t \times v)]}$$

where:

P = Ejection force (N)
S_t = Thermal contraction of the plastic across diameter d = Coefficient of thermal expansion x ΔT
ΔT = Temperature difference (°C)
d = Diameter of circle whose circumference is equal to the perimeter length of the molded part surrounding the male core (mm)
E = Elastic modulus (MPa)
A = Area of contact that shrinks onto core in the direction of ejection (mm^2)
μ = Coefficient of friction, plastic/steel
t = Thickness of molded part (mm)
v = Poisson’s ratio of the plastic

Cooling

Molds must be cooled to remove heat from the just-molded plastic part so the part can be ejected from the mold as quickly as possible. Cooling is accomplished by drilling or machining passages in the mold and circulating a heat-transfer fluid through those passages. Other than passages for cooling in the molding block or plates, the molding surfaces of the core and each cavity should also have direct cooling passages. To remove heat from the just-molded article and thus permit ejection, cooling must occur efficiently and effectively. Inefficient cooling can be very costly because cooling accounts for, on average, 70 to 80% of the cycle time.

Bore diameter for cooling channels should be drilled to accept pipes in the range of 6 to 10mm. Do not use smaller pipes unless there is a size constraint. The hoses used to interconnect passages in the mold should have the same inside diameter as the passages.

To maximize the cooling rate, the cooling fluid – water or ethylene glycol/water mixture – should flow turbulently. Turbulent flow achieves three to five times as much heat transfer as does non-turbulent flow.

The cooling rate is also affected by the material used for making the mold. A beryllium copper mold transfers twice as much heat as does a carbon steel mold, and four times as much as a stainless steel mold. This does not mean that using a beryllium copper mold will permit molding cycles four times as fast as a stainless steel mold. However, a beryllium copper mold will run some thin-wall parts significantly faster.

Beryllium copper molds are not recommended for molding thermoplastics that require elevated mold temperatures. The high thermal conductivity of the beryllium copper allows so much heat to transfer to the surroundings that it is difficult to maintain adequate heat economically. Dow Plastics offers computer-aided analysis of mold-cooling networks to help you ensure adequate and uniform cooling of your molded part.

NOTICE: Dow believes the information and recommendations contained herein to be accurate and reliable as of March 2001. However, since any assistance furnished by Dow with reference to the proper use and disposal of its products is provided without charge, and since use conditions and disposal are not within its control, Dow assumes no obligation or liability for such assistance and does not guarantee results from use of such products or other information contained herein. No warranty, express or implied, is given nor is freedom from any patent owned by Dow or others to be inferred. Information contained herein concerning laws and regulations is based on U.S. federal laws and regulations except where specific reference is made to those of other jurisdictions. Since use conditions and governmental regulations may differ from one location to another and may change with time, it is the Buyer's responsibility to determine whether Dow's products are appropriate for Buyer's use, and to assure Buyer's workplace and disposal practices are in compliance with laws, regulations, ordinances, and other governmental enactments applicable in the jurisdiction(s) having authority over Buyer's operations.