A Typical Injection Mold Design Guide

This checklist can be used as a general reference guide for injection mold design engineers. It is divided into 3 parts of a mold design process.

Part 1 - Requirements to start your mold design:

Check the injection machine where the mold is to be mounted. This will help you decide the size and structure of the mold. for ease of installation and other factors. Important notes:

Locating ring size (or other positioning method)

Nozzle size

Method of clamping (Auto or manual)

Temperature control system

Determine the number of cavities and volume requirements. This will help you decide the material that you are going to use and other mold components that you will choose for cost effective design.

Determine the gate location and size.

Determine the location where ejector pin marks are prohibited.

Part 2 - Mold base layout:

Place cavities close to the center of the mold to minimize base size and runner length.

Ensure that the molded part remains on the movable half (ejector half) upon opening of PL to facilitate proper ejection.

Waterlines should be placed as evenly as possible to the contours of the cavity.

Use support pillars underneath the cavity pockets.

Use ejector guides for molds with small ejector pins and rectangular ejector pins.

Provide eye-bolt hole for ease of mounting and dismounting.

Install mold opening prevention locks on the operator side.

Establish pry bar groove on the corners of the mold parting line to facilitate ease of mold opening during assembly and maintenance.

By this time you may ask for the mold layout approval from the customer.

Part 3 - Cavity/core details:

Check material shrinkage. Locate portions (corners) for possible significant deflection and deformation.

Maintain uniform wall thickness.

Draft angle should be within dimension tolerance.

Divide core blocks to simplify machining and provide gas vent path.

Gate, small cores, and cores with shut-off fittings are better designed as insertable components for easy modification and repair.

Watch out for possible deformation of core pins.

Position the ejector pins on the ribs and other high strength locations. Ensure ejector balance.

Detailing/part drawing: Include all parameters needed for processing -material, quantity, surface finish/texture, dimensions, tolerances and many more. Do not assume the machinist understands everything.

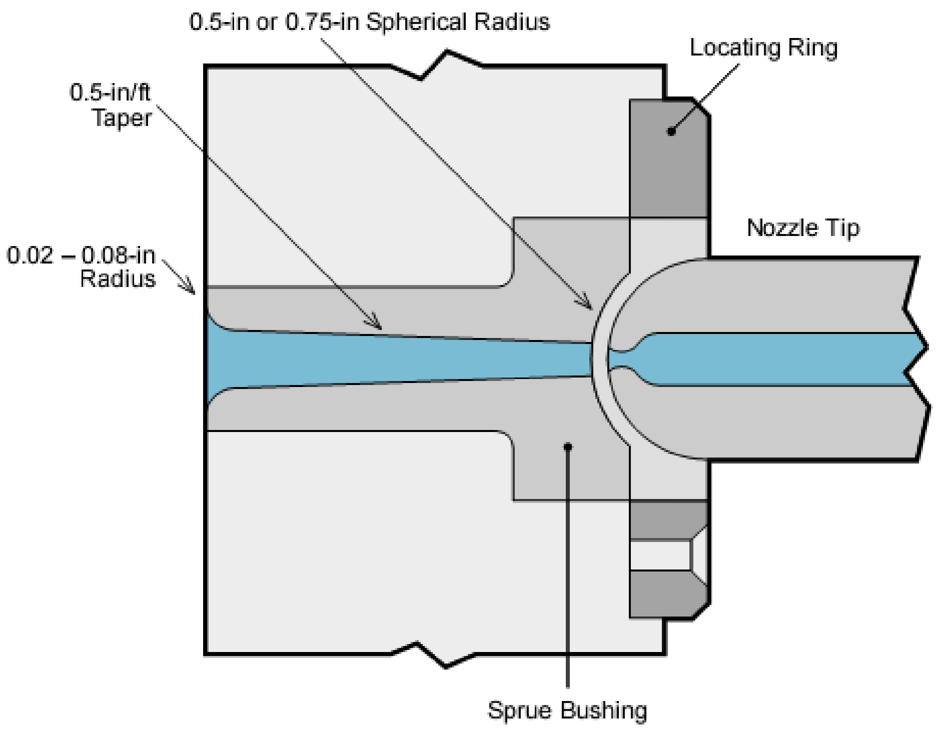
Any design change and amendments to the mold must be re-approved by the customer or mold owner.

Standard horizontal clamp presses deliver molten resin to the mold through a hole in the center of the stationary press platen. A material-delivery system — usually consisting of a sprue, runners, and gates — then leads the resin through the mold and into the cavity. These components of the material delivery system are discussed in this section.

Sprues

The sprue, oriented parallel to the press injection unit, delivers resin to the desired depth into the mold, usually the parting line. Though they can be cut directly into the mold, sprue bushings are usually purchased as off-the-shelf items and inserted into the mold (see figure 7-18). The head end of the sprue bushing comes premachined with a spherical recess — typically 0.5- or 0.75-inch radius — to receive and seal off against the rounded tip of the press injection nozzle. The sprue bushing flow-channel diameter typically tapers larger toward the parting line at a rate of 0.5 inch per foot. This eases removal of the molded sprue. The sprue orifice size, the diameter at the small end, comes standard in odd 1/32s from 5/32 to 11/32 inch. Sprue design can affect molding efficiency and ease of processing. In many molds, the greatest restriction to material flow occurs at the press nozzle tip and sprue orifice. These areas see the highest volumetric flow rate of the entire system. An excessively small sprue orifice can generate large amounts of material shear and lead to material degradation, cosmetic problems, and elevated filling pressure. The problem can be worse in the press nozzle tip because the tip orifice must be slightly smaller than the sprue orifice to avoid forming an undercut. The volumetric flow rate used during filling largely determines the correct sprue orifice size. Shot size and filling speed, as well as the flow properties of the specific resin, govern the required flow rate.

Sprue Bushing **Figure 7-18**



Sprue bushings convey the melt from the press nozzle tip to the mold parting line.

* Large parts and/or parts needing fast filling speeds require large sprue orifice diameters to avoid problems associated with excessive flow shear.
* As a general rule, amorphous resins and blends such as Makrolon polycarbonate, Lustran ABS, and Bayblend PC/ABS resins require larger sprues and runners than semicrystalline resins such as Durethan PA 6 and Pocan PBT.

The diameter at the base of the sprue increases with increasing sprue length.Standard sprue taper, typically one-half inch per foot, leads to large base diameters in long sprues. This large base diameter lengthens cooling and cycle times and also leads to regrind problems.

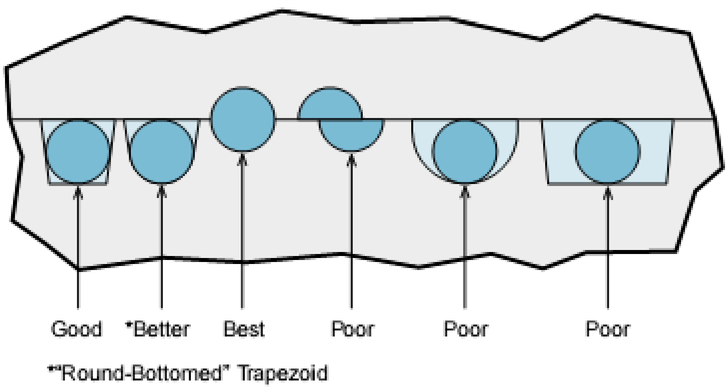
Figure 7-19 shows typical sprue sizes for Bayer amorphous resins as a function of shot size and filling time.Because the maximum shear rate in a sprue occurs at the orifice and the majority of shear heating and pressureloss takes place in the first two inches,these guidelines should apply to sprues of various lengths. Part geometry influences filling time to some extent.For example, parts with a mix of thick and thin features may need a fast filling speed to prevent premature cooling of the thin features. Other geometries may require slower filling speeds to prevent problems such as cosmetic defects or excessive clamp tonnage requirements.

Hot sprue bushings provide one solution to this problem. Hot sprue bushings have a heated flow channel that transports material along its length in molten form,eliminating or shortening the molded cold sprue. Additionally, some molds rely on extension press nozzlesthat reach deep into the mold to reduce sprue length.

Runners

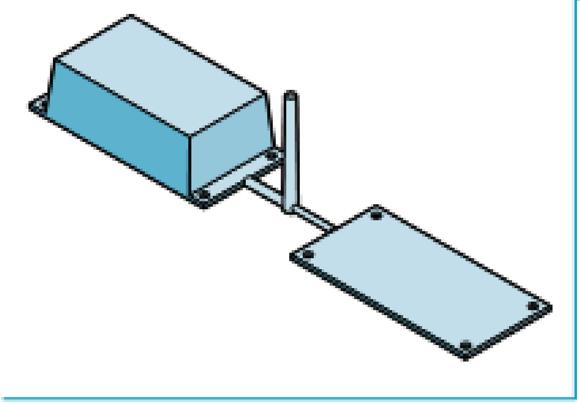
Unlike sprues, which deliver material depthwise through the center of the mold plates, runners typically transport material through channels machined into the parting line. Runner design influences part quality and molding efficiency. Overly thick runners can lengthen cycle time needlessly and increase costs associated with regrind. Conversely, thin runners can cause excessive filling pressures and related processing problems. The optimum runner design requires a balance between ease of filling,mold design feasibility, and runner volume. Material passing through the runner during mold filling forms a frozen wall layer as the mold steel draws heat from the melt. This layer restricts the flow channel and increases the pressure drop through the runner. Round cross-section runners minimize contact with the mold surface and generate the smallest per-centage of frozen layer cross-sectional area. As runner designs deviate from round, they become less efficient (see figure 7-20). Round runners require machining in both halves of the mold, increasing the potential for mismatch and flow restriction. A good alternative, the “round-bottomed” trapezoid, requires machining in just one mold half. Essentially a round cross section with sides tapered by five degrees for ejection, this design is nearly as efficient as the full-round design. The runner system often accounts for more than 40% of the pressure required to fill the mold. Because much of this pressure drop can be attributed to runner length, optimize the route to each gate to minimize runner length. For example, replace cornered paths with diagonals or reorient the cavity to shorten the runner.

Runner Cross Sections Figure 7-20



Full round runners provide the most efficient flow.

Family Mold Figure 7-23



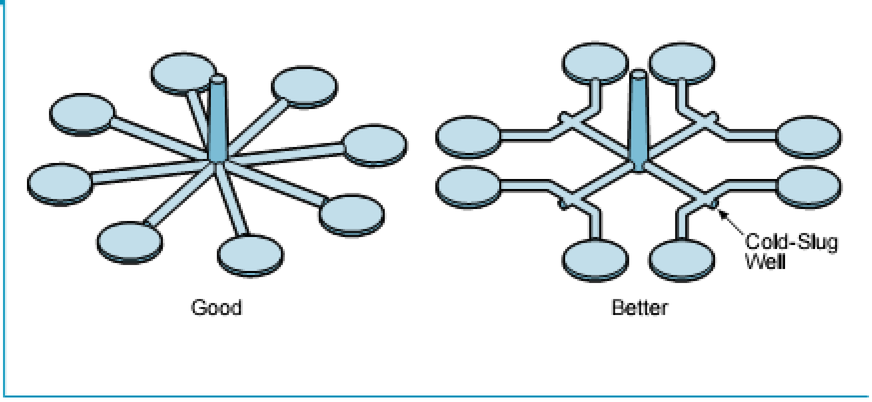
The runner diameter feeding the smaller part was reduced to

Figure 7-23 Family Mold

The runner diameter feeding the smaller part was reduced to balance filling.

Runners for multicavity molds require special attention. Runners for family molds, molds producing different parts of an assembly in the same shot, should be designed so that all parts finish filling at the same time. This reduces over- packing and/or flash formation in the cavities that fill first, leading to less shrinkage variation and fewer part-quality problems. Consider computerized mold- filling analysis to adjust gate locations and/or runner section lengths and diam-eters to achieve balanced flowto each cavity (see figure 7-23). The same computer techniques balance flow within multi-gated parts. Molds producing multiples of the same part should also provide balanced flow to the ends of each cavity. Naturally balanced runners provide an equal flow distance from the press nozzle to the gate on each cavity. Spoked-runner designs (see figure 7-24) work well for tight clusters of small cavities. However they become less efficient as cavity spacing increases because of cavity number or size.

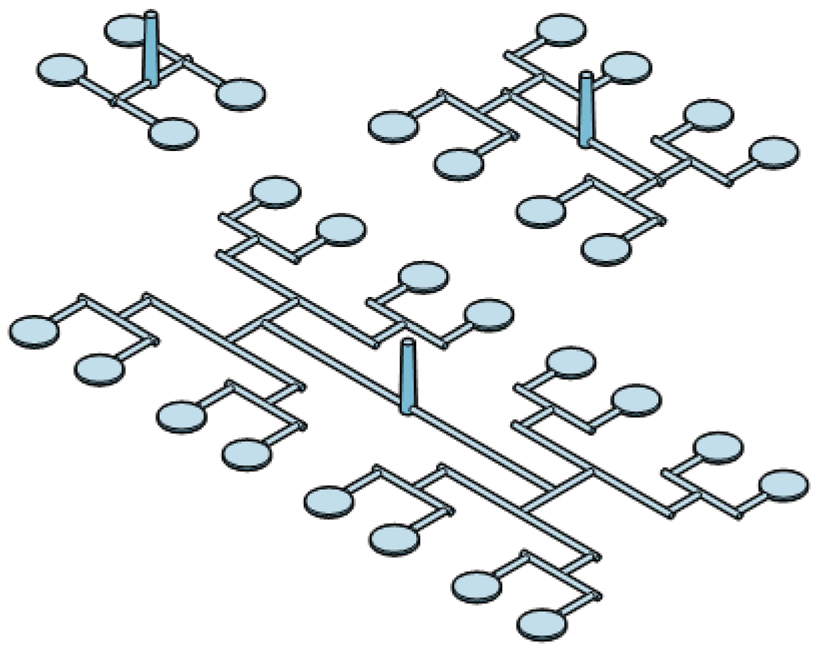
Figure 7-24 Spoked Runners



The spoked runner on the right provides a cold slug well at the end of each primary runner branch.

Often, it makes more sense to orient cavities in rows rather than circles. Rows of cavities generally have branched runners consisting of a primary main feed channel and a network of secondary or tertiary runners to feed each cavity. To be naturally balanced, the flow path to each cavity must be of equal length and make the same number and type of turns and splits. This generally limits cavity number to an integer power of two — 2, 4, 8, 16, 32, etc. —as shown in figure 7-25. Generally, the runner diameter decreases after each split in response to the decreased number of cavities sharing that runner segment. Assuming a constant flow rate feeding the mold, the flow-front velocity in the cavity halves after each split. The molding press flow-rate performance may limit the number of cavities that can be simultaneously molded if the press cannot maintain an adequate flow-front velocity.

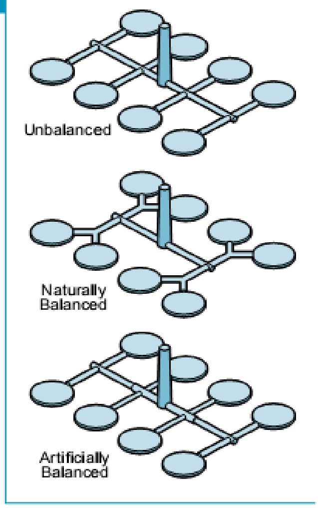
Naturally Balanced Runners **Figure 7-25**



Naturally balanced runners for cavities in two rows.

Artificially balanced runners provide balanced filling and can greatly reduce runner volume. Artificially balanced designs usually adjust runner-segment diameters to compensate for differences in runner flow length. For instance, in ladder runners, the most common artificially balanced runner design, a primary runner feeds two rows of cavities through equal-length secondary runners. The diameters of these secondary runners are made progressively smaller for the cavities with shortest runner flow distance (see figure 7-26). These designs require enough secondary runner length to flow balance using reasonable runner diameters.

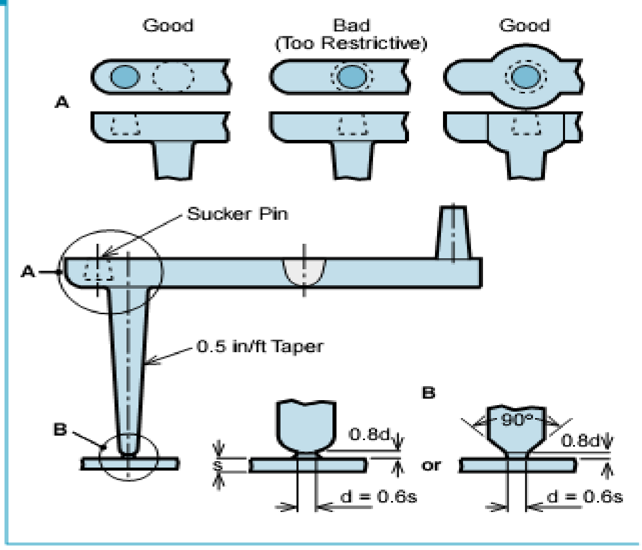
Figure 7-26 Runner Balancing



The artificially balanced runner achieves flow balance by adjusting runner diameters instead of by maintaining uniform runner length.

As a general rule, secondary runner length should be no less than 1/5 the flow distance from the inboard secondary/primary runner junction to the gates on the outboard cavities. Runners for three-plate molds initially convey material along the runner-split parting line and then burrow perpendicularly through the middle plate to the cavity parting line. Tapered drops typically project from the main runner to pinpoint gates on the part surface. To ease removal from the mold, these drops taper smaller toward the gate at a rate of about 0.5 inch per foot. Avoid long drops because the taper can lead to excessive thickness at the runner junction or flow restriction at the thin end. Three-plate runners usually require sucker pins or some other feature to old the runner on the stripper plate until the drops clear the center plate during mold opening. Be sure these features do not restrict flow. See figure 7-27 for three-plate runner and gate-design guidelines.

Figure 7-27 Three-Plate Runner



Three-plate runner system guidelines.

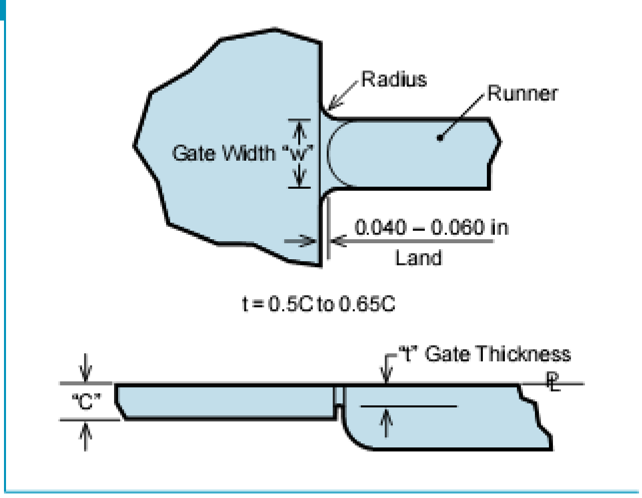
Gates

Except for special cases, such as sprue-gated systems which have no runner sections, gates connect the runner to the part. Gates perform two major functions, both of which require the thickness to be less than the runner and part wall. First, gates freeze-off and prevent pressurized material in the cavity from backing through the gate after the packing and holding phases of injection. Applied pressure from the press injection unit can stop earlier in the cycle, before the part or runner system solidifies, saving energy and press wear-and-tear. Secondly, gates provide a reduced-thickness area for easier separation of the part from the runner system. A variety of gate designs feed directly into the parting line. The common edge gate(see figure 7-28) typically projects from the end of the runner and feeds the part via a rectangular gate opening. When designing edge gates, limit the land length, the distance from the end or edge of the runner to the part edge, to no more than 0.060 inch for Bayer thermoplastics. Edge gates generate less flow shear and consume less pressure than most self degating designs. They are therefore preferred for shear-sensitive materials, high-viscosity materials, highly cosmetic applications, and large-volume parts.

Fan gates and chisel gates, variations of the edge gate, flare wider from the runner (see figure 7-29) to increase the gate width. Chisel gates can provide better packing and cosmetics than standard edge gates on some thick-walled parts. Like the standard edge gate, the land length for fan gates should not exceed 0.060 inch at the narrowest point. Chisel gates taper from the runner to the part edge with little or no straight land area.

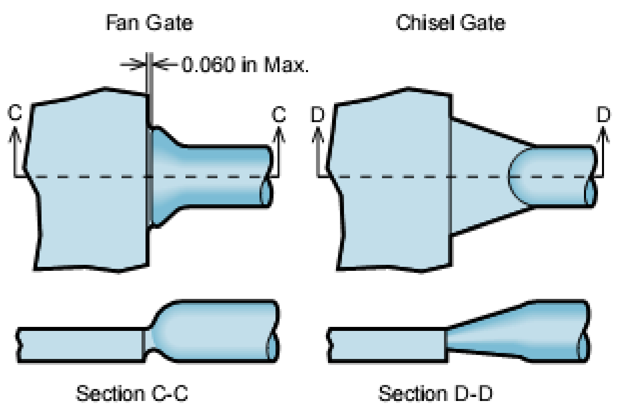
Edge gates can also extend to tabs (see figure 7-30) that are removed after molding or hidden in assembly. These tab gates allow quick removal of the gate without concern about gate appearance.

Figure 7-28 **Common Edge Gate**



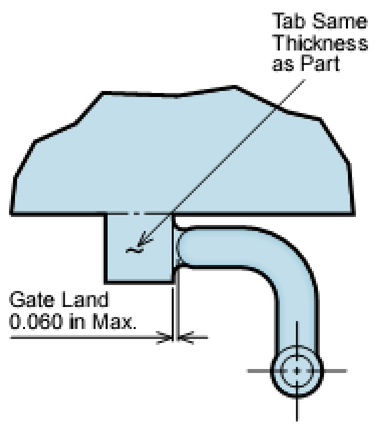
Common edge-gate guidelines.

Figure 7-29 **Variations of the Edge Gate**



Fan gates and chisel gates can provide better cosmetics in some applications.

Figure 7-30 **Tab Gate**



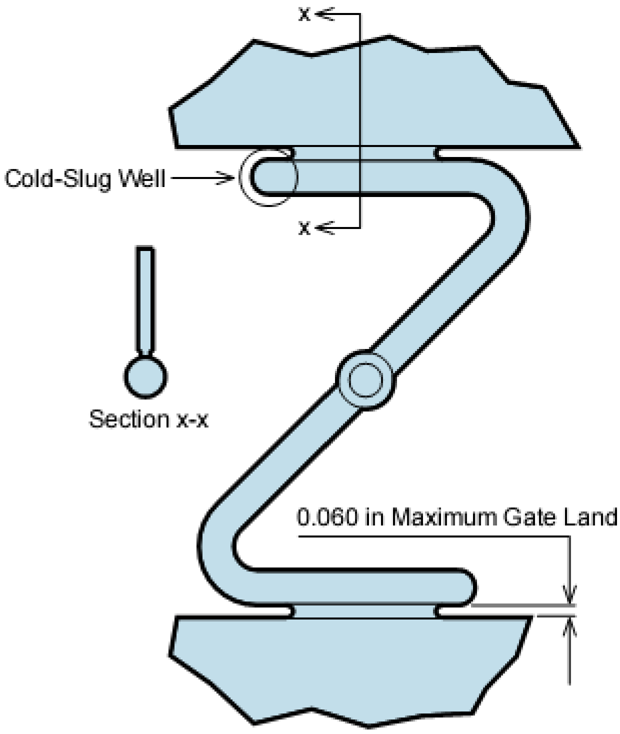
The gate tab can be hidden in the assembly or trimmed off after molding.

Edge gates may also extend from the side of a runner oriented parallel to the part edge (see figure7-31). This design, coupled with a “Z”-style runner, tends to reduce gate blush by providing uniform flow along the width of the gate and a cold-slug well at the end of the runner. To hide the large gate vestige left by large edge gates, the gate can extend under the edge as shown in figure 7-32.

Because they extend under the mold parting surfaces, tunnel gates can reach surfaces or features that are not located on the parting line. The gates typically feed surfaces oriented perpendicular to the mold face. Depending upon their design, they degate during ejection or mold opening (see figures 7-33 and 7-34). Tunnel gates that degate during mold opening often require a sucker pin or a feature similar to a sprue puller to hold the runner on the ejector half of the mold. The runner must flex for the gate to clear the undercut in the mold steel. The gate may break or lock in the mold if the runner is too stiff or if the ejector pin is too close to the gate. Normally, the ejector pin should be at least two runner diameters away from the base of the gate. The orifice edge closest to the parting line must remain sharp to shear the gate cleanly. When molding abrasive materials such as those filled with glass or mineral, make the gate of hardened or specially treated mold steel to reduce wear. Also, consider fabricating the gate on an insert for easy replacement. The drop angle and conical angle must be large enough to facilitate easy ejection (see figure 7-35). Stiff materials, glass-filled grades for example, generally require drop angles and conical angles at the high side of the range shown in the figure. The modified-tunnel gate design (see figure 7-36) maintains a large flow diameter up to the gate shear-off point to reduce pressure loss and excessive shear heating.

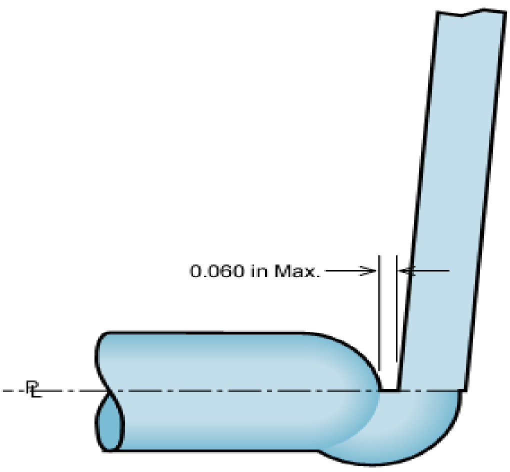
Curved-tunnel gates permit gating into the underside of surfaces that are oriented parallel to the parting plane (see figure 7-37). Unlike mold fabrication for conventional tunnel gates, the curved, undercut shape of this design must be machined or EDM burned on the surface of a split gate insert. The curved gate must uncurl as the runner advances on guided posts during ejection. This gate design works well for unfilled materials that remain somewhat flexible at ejection temperature such as Makrolon PC, Lustran ABS, and amorphous blends such as Bay blend and Makroblend resins. Avoid this gate for filled materials, brittle materials, or materials with very high stiffness. See figures 7-38 and 7-39 for curved-tunnel gate design guidelines.

"Z" Runner **Figure 7-31**



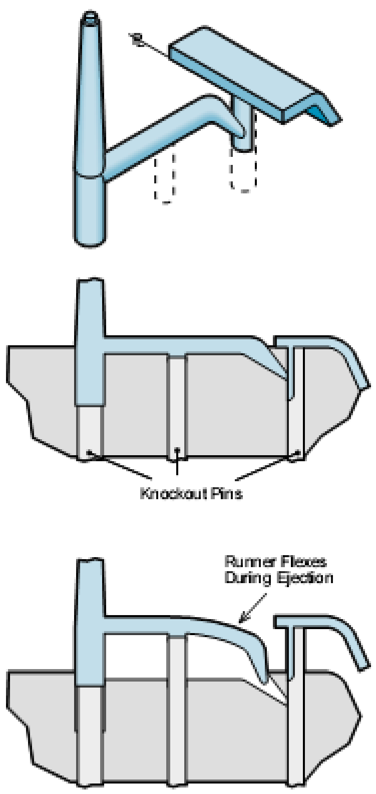
Edge gate from the side of a "Z" runner.

Gate Under the Edge **Figure 7-32**



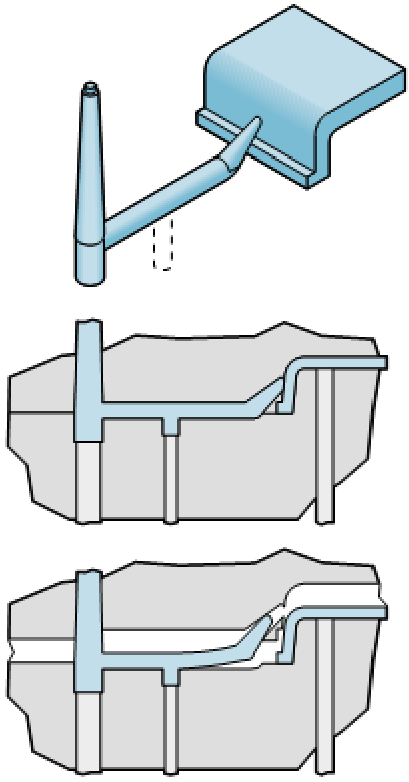
This gate can be trimmed without leaving a gate mark on the cosmetic part surface.

Knockout-Pin Gate **Figure 7-33**



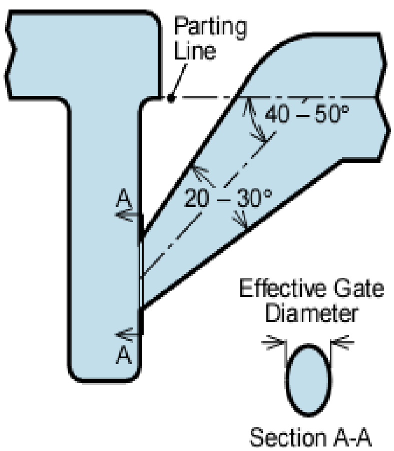
Tunnel gates that extend below the parting line on the ejector side of the mold degate during ejection.

Stationary-Side Tunnel Gate **Figure 7-34**



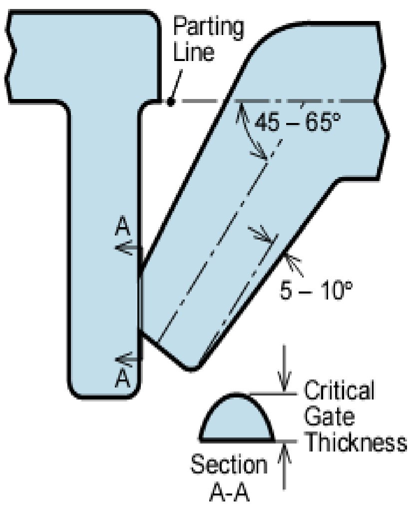
Tunnel gates into non-ejector side of the mold degate and separate from the part during mold opening.

Tunnel-Gate Configuration **Figure 7-35**



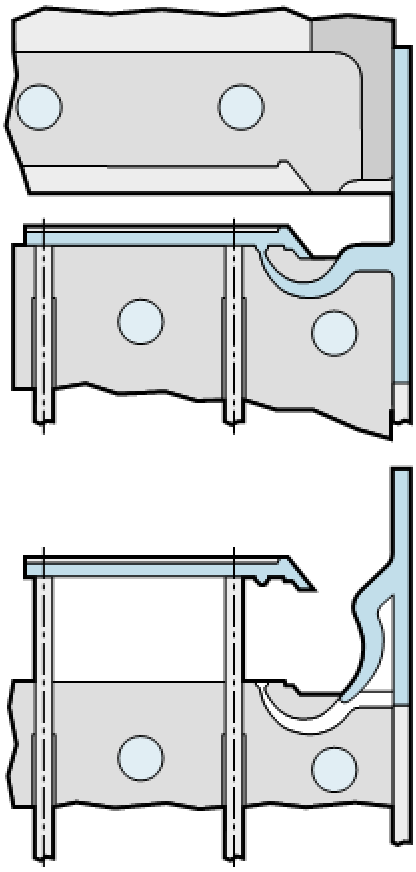
Standard tunnel-gate guidelines.

Modified Tunnel-Gate Configuration **Figure 7-36**



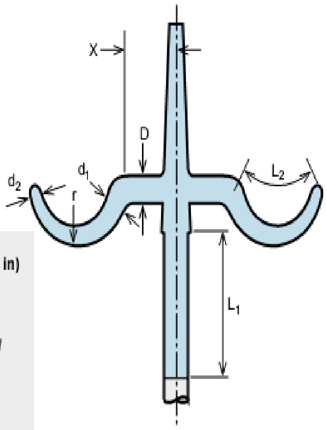
Modified tunnel-gate guidelines.

Curved-Tunnel Gate **Figure 7-37**



Curved-tunnel gates can reach past the finished edge to the underside of surfaces oriented parallel to the parting plane.

Curved-Tunnel-Gate Guidelines **Figure 7-38**



$\frac{X}{D} \geq \frac{2.5}{1}$ or Min. 15 mm (0.600 in)

D = Approx. 4 to 6 mm (0.160 to 0.235 in)

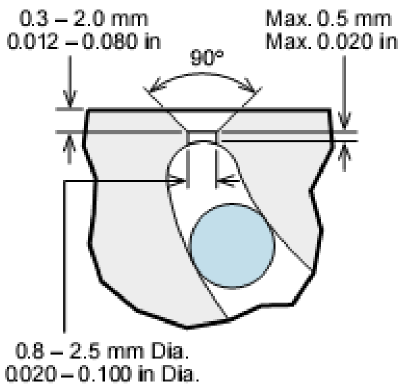
$d_1 \leq D$ (Normally 4 to 6 mm / 0.150 to 0.235 in)

r = 2.5 to 3 x d_1

d_1 to d_2 Equals a Taper of 3° to 5° Incl.

$L_1 \geq L_2$

Figure 7-39 **Curved-Tunnel-Gate Design Guidelines**

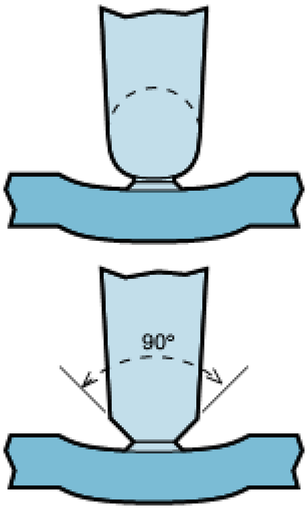


The curved tunnel gate needs a well-defined break-off point for clean degating.

Pinpoint gates feed directly into part surfaces lying parallel to the mold parting plane. On the ends of three-plate runner drops, multiple pinpoint gates can help reduce flow length on large parts and allow gating into areas that are inaccessible from the part perimeter. For clean degating, the gate design must provide a positive break-off point (see figure 7-40) to minimize gate vestige. Set in recesses or hidden under labels, properly designed and maintained pinpoint gates seldom require trimming. Because gate size must also be kept small, typically less than a 0.080-inch diameter, pinpoint gates may not providesufficient packing for parts with thick wall sections.

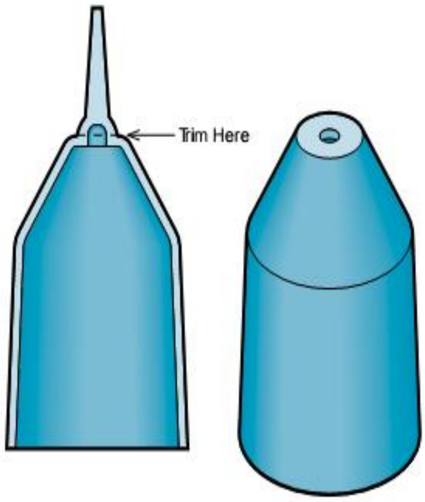
Parts with holes in the center such as filter bowls, gears, and fans often use the “filter-bowl”gate design to provide symmetrical filling without knit lines. Typically, the gate extends directly from a sprue and feeds the cavity through a continuous gate into the edge of the hole (see figure 7-41). Degating involves trimming away the sprue and conical gate section flush with the outer surface.

Figure 7-40 **Pinpoint Gate**



Both of these pinpoint gate designs provide a well-defined break-off point for clean degating. Design permitting, pinpoint gates should be placed in recessed gate wells to accommodate gate vestige.

Figure 7-41 **Filter-Bowl Gate**



Typical filter-bowl gate avoids knitlines and provides even flow around the core.