HANSER

Gastrow Injection Molds

Peter Unger

130 Proven Designs

ISBN 3-446-40592-5

1 Principles of Mold Design

General Remarks

In an article reporting on the Ninth Euromold Fair, we read, [1] "Mold and die making is alive and well in Germany." The innovative strength of the field speaks for this claim. Even if production, and the know-how that goes with it, are being shifted out of the country, the truth is, "Much more significant for securing long-term perspectives are: continued technological progress with respect to production-cost cutting and product functionality, as well as unbending and far-sighted training to motivate the next generation." [2] From its very inception, the "Gastrow", being a reference work and source of ideas, has been dedicated to the goal of disseminating knowledge. This new edition aims to do so more as a collection of examples to help find design solutions. Computer methods, i.e., CAD, can at best supplement and optimize a design concept with, for example, rheological, thermal, and mechanical mold configuration, but, as all experience shows, cannot replace it. Moreover, it remains the case that the results of CAD have to be critically evaluated – a task that requires sophistication and practical experience. Thus it remains common practice in the production of precision-made injection molded parts to build a test mold, or at least a test cavity, in order to optimize dimensional stability, for example, and adapt to requirements (in several steps). CAD results often indicate only the determination for shrinkage (warping), a characteristic of molded parts, especially those made from semi-crystalline polymers, that is quite diffcult to quantify. Even so, development time and costs can undoubtedly be reduced by suitable computer methods. For information on applying computer methods, the reader should consult the relevant literature.

There may be no *objective* rule dictating *the* right way to classify anything, but there is *a* right way, namely to organize the subject matter so thoroughly that all phenomena are covered and so clearly that the mind receives a distinct overview of the total. Of course, time and experience cause us to see the phenomena differently, expand and alter the things to be classified and, in so doing, provide an additional pathway of understanding that does not always sit well with a classification system rooted in the past. In this respect, injection molds are no different from anything else: some of the terminology is theoretically clear, some does not become clear unless one knows when and where it came from. Since engineering is the practical offspring of science, historical example is a major source of knowledge as inspiration for the engineer, helping to bridge the gap between theory and practice. For the mold designer working on a problem, consulting previous practice can save time and locate the areas that require real work, i.e., innovation. He can see how others have faced and solved similar problems, while he can evaluate their results and create something even better – instead of "reinventing the typewriter". One basic requirement to be met by every mold intended to run on an automatic injection molding machine is this: the molded part has to be ejected automatically and not require subsequent finishing (degating, machining to final dimensions, etc.)

For practical reasons, injection molds are best classified according to both the major design features of the molds themselves and the molding-operational features of the molded parts. These include the

- type of gating/runner system and means of separation
- type of ejection system for molded parts
- presence or absence of external or internal undercuts on the part to be molded
- the manner in which the molded part is to be released.

The final mold design cannot be prepared until the part design has been specified and all requirements affecting the design of the mold have been clarified.

1.1 Types of Injection Molds

The DIN ISO standard 12165, "Components for Compression, Injection, and Compression-Injection Molds" classifies molds on the basis of the following criteria:

- standard molds (two-plate molds)
- split-cavity molds (split-follower molds)
- stripper plate molds
- three-plate molds
- stack molds
- hot runner molds

Generally, injection molds are used for processing

- thermoplastics
- thermosets
- elastomers

There are also cold runner molds for runnerless processing of thermosetting resins in analogy to the hot runner molds used for processing thermoplastic compounds and elastomers.

Sometimes runners cannot be located in the mold parting plane, or each part in a multi-cavity mold has to be center-gated. In such cases, either a second parting line (three-plate mold) is required to remove the solidified runner, or the melt has to be fed through a hot runner system. In stack molds, two or more molds are mounted back-to-back in the line of closing, but without multiplying the required holding force. The prerequisite for such solutions is large numbers of relatively simple, e.g., flat molded parts, and their attractiveness comes from reduced production costs. Today's stack molds are exclusively equipped with hot runner systems that have

to meet strict requirements, especially those involving thermal homogeneity.
For ejecting molded parts, mainly ejector pins are used. These often serve, in addition, to transfer heat and vent the cavity. Venting has become a major problem since electrical discharge machining (EDM) has become state-of-the-art. Whereas cavities used to be "built up" from several components, thus providing for effective venting at the respective parting planes, EDM has, in many cases, enabled the production of cavities from a single massive block. Special care must be taken to ensure that the melt displaces all air, and that no air remains trapped in the molded part – an especially sensitive issue. Poor ventilation can lead to deposits on cavity surfaces, and to the formation of burn spots (so-called "diesel effect") and even to corrosion problems. The size of venting gaps is essentially determined by the melt viscosity. They are generally on the order of 1/100 mm to approx. 2/100 mm wide. When extremely easy flowing melts are to be processed, vents have to measure in thousandths of a millimeter to ensure that no flash is generated. It must be noted that effective heat control is generally not possible in regions where a vent is provided. As for venting elements – such as venting inserts made from sinter metal – they require regular servicing due to time-factored pore-clogging that varies with the material being processed. Care must be taken when positioning venting elements in the cavity.
Moving mold components have to be guided and centered. The guidance provided by the tiebars for the moving platen of an injection molding machine can be considered as rough alignment at best. "Internal alignment" within the injection mold is necessary in every instance.
Tool steels are the preferred material for injection molds. The selection of materials should be very careful and based on the resins to be processed. Some of the properties required of tool steels are

- high wear resistance
- high corrosion resistance
- good dimensional stability (see also Section 1.9)

Molds made from aluminum alloys are also gaining in popularity, see also Section 1.10.3.1.

1.2 Types of Runners and Gates

1.2.1 Solidifying Systems

According to DIN 24450, a distinction is made between the terms

- 'runner' (also termed 'sprue') meaning that part of the (injection molding) shot that is removed from the molded part
- 'runner' meaning the channel that plasticated melt passes through from its point of entry into the mold up the gate and
- 'gate' meaning the cross-section of the runner system at the point where it feeds into the mold cavity.

The flow path of the melt into the cavity should be as short as possible in order to minimize pressure and heat losses. The type and location of runner/gate are important for:

- economical production
- properties of the molded part
- tolerances
- weld lines
- magnitude of molded-in stresses, etc.

The following list provides an overview of the most commonly encountered types of solidifying runner systems and gates.

- Sprus (Fig. 1.1)

are generally used when the parts have relatively thick walls or when highly viscous melts require gentle processing. The sprue has to be removed mechanically from the molded part after ejection. Appropriate sprue bushes are available as standard units in various versions, for example, with twist locks, temperature control, etc., see also ISO 10072. Due to their large flow diameters, conventional sprues exhibit minimal pressure loss. However, it must be taken into consideration that a too-large sprue can determine the cycle time. Thus maximum diameter ought not to exceed part wall-thickness plus approx. 1.5 mm. If temperature-controlled (cooled) sprue bushes are used, this value may be exceeded. Conventional sprues offer optimum holding time in the injection molding process. To prevent sink marks or non-uniform gloss, suffcient (separate) cooling power should be provided at a distance from the gate.

- Pinpoint (Fig. 1.2)

In contrast to the sprue, the pinpoint gate is generally separated from the molded part automatically. If gate vestige presents a problem, the gate d_1 can be located in a lens-shaped depression on the surface of the molded part. Commercially available pneumatic nozzles are also used for automatic ejection of a runner with pinpoint gate. Pinpoint gating has been especially successful in applications for small

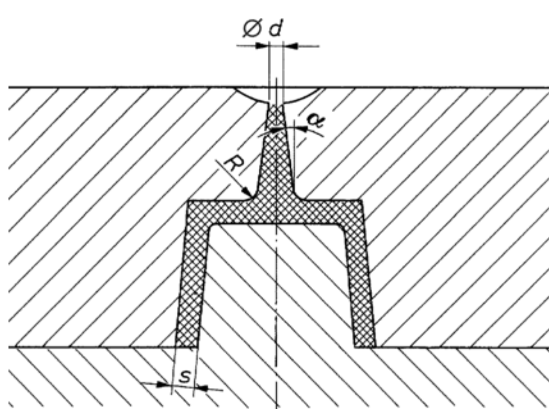

Figure 1.1 Conventional sprue
α = draft, s = wallthickness, d = sprue(diameter), $d \leq 1.5 + 5$ [mm]; $d \geq 0.5$ mm; 15 [mm]

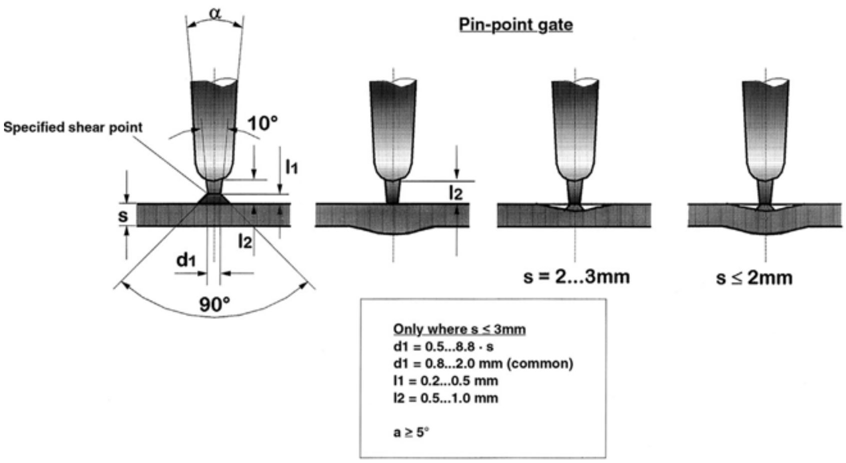

Figure 1.2 Pinpoint gate
(Courtesy: Ticona)

and/or thin-walled molded parts. At separation, however, drool has been a problem with certain polymers and premature solidification of the pin gate may make it diffcult to optimize holding time.

- Diaphragm gate (Fig. 1.3a)

The diaphragm is useful for producing, for instance, bearing bushings with the highest possible degree of concentricity and avoidance of weld lines. Having to remove the gate by means of subsequent machining is a disadvantage, as is one-sided support for the core. The diaphragm, Fig. 1.3, encourages jetting which, however, can be controlled by varying the injection rate so as to create a swelling material flow. Weld lines can be avoided with this type of gating.

- Disk gate (Fig. 1.3b)

This is used preferably for internal gating of cylindrical parts in order to eliminate disturbing weld lines. With fibrous reinforcements such as glass fibers, for instance, the disk gate can aggravate the tendency for distortion. The disk gate also must be removed subsequent to part ejection.

- Film gate (Fig. 1.4)

To obtain flat molded parts with few molded-in stresses and little tendency to warp, a film gate over the entire width of the molded part is useful in providing a uniform flow front. A certain tendency of the melt to advance faster in the vicinity of the sprue can be offset by correcting the cross-section of the gate. In single-cavity molds, however, the offset gate location can cause the mold to open on one side, with subsequent formation of flash. The film gate is usually trimmed off the part after ejection, but this generally does not impair automatic operation. Immediately following removal, i.e., in the "first heat", the film gate should be separated mechanically, in order to ensure that the molded part does not warp in the gate area (since the gate's wall thickness is less than that of the molded part, greater and smaller differences in shrinkage may arise and encourage warping).

- Submarine gate (Fig. 1.5)

Depending on the arrangement, this type of gate is trimmed off the molded part during mold opening or directly on ejection at a specified cutting edge. The submarine gate is especially useful when gating parts laterally. Aside from potential problems due to premature solidification, submarine gates can have very small cross sections, leaving virtually no trace on the molded part. With abrasive molding compounds, increased wear of the cutting edge in particular is to be expected. This may lead to problems with automatic degating.

Runner systems should be designed to provide the shortest possible flow paths, avoiding unnecessary changes in direction, while achieving simultaneous and uniform cavity filling regardless of position in multi-cavity molds (assuming identical cavities) and ensuring identical duration of holding pressure for each cavity.

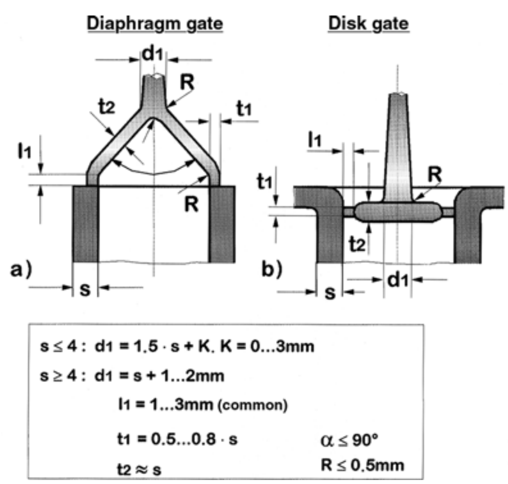

Figure 1.3 Diaphragm (a) and disk (b) gate
(Courtesy: Ticona)

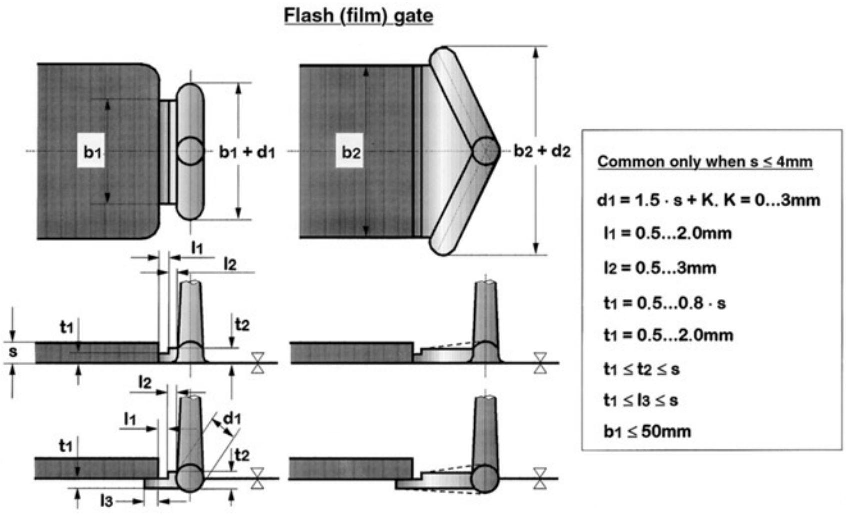

Figure 1.4 Flash or film gate
(Courtesy: Ticona)

For thermoplastics with a high modulus of elasticity (brittle-hard demolding behavior), the angle on the cutting edge has to be relatively small, e.g., $\alpha = 30°$. For thermoplastics with a low modulus of elasticity (viscoplastic removal behavior), *curved submarine gates* have proven successful, Figs. 1.6 and 1.7. In such molds, the gate is separated at a specified point, as with pinpoint gating. Using this type of gating, several submarine gates with short distances in between can produce approximately the same flow pattern as when a film gate is used, but with the considerable advantage that the gate is separated automatically from the molded part, Fig. 1.6. Certain peculiarities of this type of gate have to be kept in mind. For example, the runner must have a lengthened guide and, if necessary, a specified shear point, Fig. 1.6 (right segment), in order to ensure trouble-free separation and removal of the sprue. Replaceable runner inserts are available commercially. One-piece inserts manufactured by the MIM process, e.g., made from Catamold (BASF), are regularly available in round or angular versions with gate diameters between 0.5 and 3 mm [3]. An interesting new development is the swirl-flow insert, since it can be used to gate molded parts "around corners", Fig. 1.8. It is a good idea to provide for separate temperature control as close to the gate inserts as possible.

- Rectangular gate (Fig. 1.9)

Thanks to lower pressure losses and, in consequence, improved pressure transfer, the rectangular gate is sometimes an attractive alternative to point

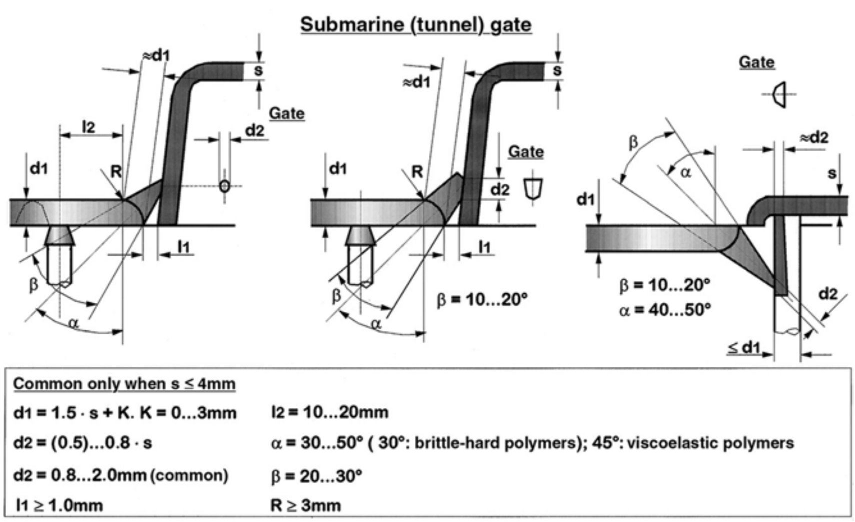

Figure 1.5 Submarine gate
(Courtesy: Ticona)