Injection Mold Design Guidelines

THIRD IN A SERIES

Maximizing Performance Using Copper Alloys

By Dr. Paul Engelmann and Bob Dealey
for the Mold Marketing Task Group of the Copper Development Association

The Injection Mold Cavity
A mold cavity forms the exterior of the plastic part and almost always is located on the "A" side of the mold. However, there are situations where the cavity is located on the opposite side of the parting line or occasions where the injection-molded part is symmetrical and the cavity is on both sides of the parting line. In these instances it is called a "B" side cavity.

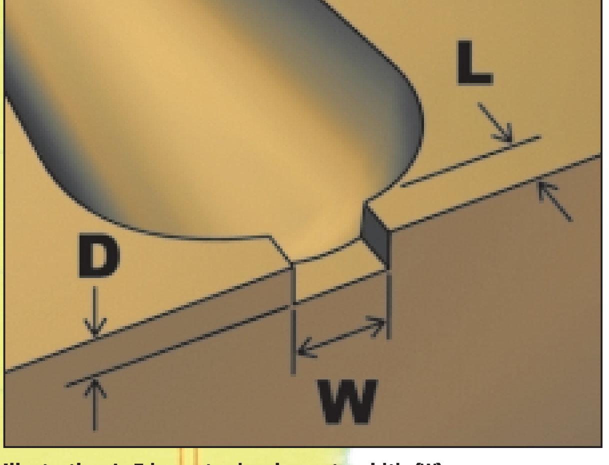

Illustration A: Edge gate showing gate width (W), depth (D) and gate land (L).

Cavity Inserts
Molds containing more that one cavity, multiple impression molds, are usually constructed individually, or if small, ganged into cavity blocks, and inserted into the mold "A" or cavity plate. Separate inserting allows for ease of manufacturing the cavity from a variety of mold materials and makes replacement easier should damage ever occur. Round inserts are a natural, as round insert pockets are easy to machine with great accuracy into mold plates. The round insert is an ideal application for "surround" cooling of the insert. "O" rings are used to seal the water channels and prevent coolant leakage. They should be designed to be placed in compression and not in shear, for ease of insert installation and leak free operation.

Mono Block Construction
Frequently in low run single impression and/or large molds the cavity is machined directly into the "A" plate. This type of mold construction is referred to as "Mono-Block" construction and eliminates the step of machining an insert pocket into the plate. Additionally, it offers a very rigid type mold construction with opportunities for excellent cooling channels surrounding the cavity.

Cavity Cooling
Cooling channels of appropriate diameter or channel size should be incorporated into the mold cavity block. Placement of these channels should follow similar recommendations made for the core of the mold. The coolant channel should be placed about two times the diameter away from the cavity with a pitch of three to five times the diameter. While it is common practice to run the cavity warmer than the core for aesthetic reasons, the best running molds are those where even surface temperatures and good temperature control can be maintained.

To insure turbulent water flow in mold cooling circuits a mold cooling analysis is conducted prior to mold building. This analysis checks and determines place-

SPECIAL ADVERTISING SECTION

Issue date, Folio #, Date, Time
MODERN PLASTICS,
BLACK CYAN MAGENTA YELLOW
PMS ID
5% 25% 50% 75% 95%
RHP

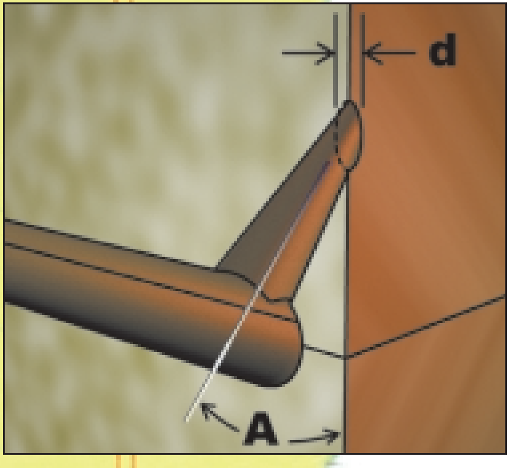

Illustration B: Submarine or tunnel gate, diameter and angle of cone illustrated.

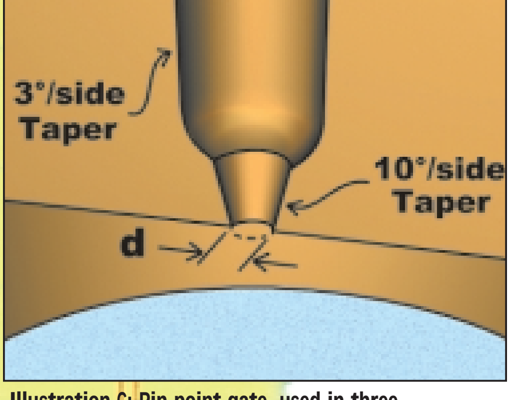

Illustration C: Pin point gate, used in three plate molds.

ment, number and size of coolant channels required. After the mold is built coolant flow can be measured to determine if adequate flow rates are being maintained. The following table lists pipe size and minimum amount of flow in gallons per minute which guarantees turbulent flow.

Nominal Pipe Size (NPT)	Channel Size (diameter)	Min. Flow (Gal/Min.)
1/16	.250	0.34
1/8	.313	0.45
1/4	.438	0.56
3/8	.562	0.75
1/2	.688	1.30

Core Inserts in Cavities

Holes in the part and other male core detail formed from the cavity side are normally achieved with core inserts in the cavity. Consideration has to be made for removal of heat from these components. Parts like television backs with air circulation slots are idea candidates for inserting high thermal conductivity copper alloys, C17200, C17510 or C18000 into the cavity, as are other hard to install coolant channels in other sections of the molds.

A pure mold cavity for an item like a drinking glass, where the plastic shrinks away from the cavity, only has to remove about 25 to 33% of the heat from the plastic. Plastic parts with contoured configurations such as toys sometimes are molded with 50% of the heat removed through the cavity. These parts benefit from the high thermal conductivity of the copper alloys and the more even mold surface temperatures offered.

Mold Base Materials

Additionally the mold plate can be build from the material most ideally suited for the type and amount of cycles the mold will run. The mold plates and/or mold base are normally built from 1030 plain carbon steel, Number 1steel for prototypes or very short runs. Number 2 steel, 4130-4140, is most often used for medium run molds. P-20 or Number 3 steel is used for long run molds and when part detail is machined into the mold plate. When a heat-treated mold plate is required, some cavity detail can be formed on the mold plate and the cavity built up with laminations, H-13 is a logical choice for these high run molds. Another material used for high volume and long running molds or with corrosive plastics is type 420F stainless steel.

Cavity Materials

When high thermal conductivity is required, either to more rapidly cool the plastic and promote faster cycles or to maintain a more even surface temperature for better dimensional consistency, the copper alloys designated C17200, C17510 or C18000 are used.

Gate Placement, Types and Size

Gate placement, normally in the cavity, is critical to the plastic component ultimate aesthetic and physical properties. The ideal location will, to a degree, dictate the type and size of the gate used for the mold. The best gate location, along with the type and size of the gate, is one of the early and difficult mold design decisions that must be made correctly to insure a good running mold.

In injection molding the more commonly used gates are the edge, submarine (tunnel) and pinpoint gate, which constitutes about 65% of molds built and which we will elaborate on. However, some parts are more ideally suited to different gate configurations including runnerless molding system drops, fan, tab, sprue, ring, diaphragm, flash and post gates.

Edge Gates

The edge gate is the most common type of gate on conventional molds. (Illustration A) Typically, the gate depth (d) is 50 to 80% of the wall that it is connected to. The depth of the gate controls gate freeze off and is the most critical dimension in determining pressure loss. Width of the gate (w) is generally two to four times the depth, depending upon the volume of plastic required to fill the part. The gate land (l) should be short to avoid large pressure losses. Additionally, a short gate land will assist in breaking or degating the part from the runner.

Submarine Gate
The submarine (frequently referred to as a tunnel or sub) gate is a popular choice on conventional molds when an automatic method of separating the part from the runner system is desired. The diameter of the gate (Illustration B) is generally 50 to 70% of the wall section, with 60% a common choice. The angle of the cone is normally 30°. However, with stiffer materials it frequently is less. Care must be taken so as not to place the cone too close to the cavity thereby avoiding thin cavity material sections and premature failure. Flexible plastics allow the angle to increase, as the plastic will still pull from the cone. Most submarine gates are placed to enter on the cavity side of the mold. In these situations, ejector pins with pullers are incorporated in the runner system to hold the runner during gate separating and then eject the runner.

Pin Point Gate
Pin point gates (top gating) are used in modified and three plate molds. Typically, and almost without exception, round parts like closures and caps have the gate located in the center of the part. The cone, connecting the runner system with the gate starts with a full radius at the end and typically has a 2°-3° angle to assist in extraction with the assistance of a sucker pin. The gate portion tapers down at about a 10° angle. The intersection of the gate to the cavity must be sharp providing for a good break off. Generally, the amount of gate projection from a pin point gate is one-half the gate diameter. The gate diameter (Illustration C) is 30 to 60% of the wall that it is connected to. The ideal gate diameter is a function of the flow properties of the plastic; the easier the flow the smaller the gate diameter. The harder the flow, the larger the diameter. Obviously, the smaller the gate diameter the better the cosmetics on the part and the smaller the gate vestige.

The ideal location for the gate is listed below. Unfortunately, sometimes trade offs must be made and not every criteria can be satisfied. Therefore, a decision has to be made as what is the best compromise in gate placement. 1. Plastic must have the ability to fill the entire part without using extreme processing conditions and maximum injection pressures. (Multiple gates may be required on some parts) 2.Material, flowing from the gate, will push gasses toward the parting line or other areas where they can be effectively vented and will not entrap gas. 3. Plastic will flow into the thickest section of the part and will flow from thicker to thinner sections. 4. Flow to all points on the part will be of equal distance and the part will not experience areas of over packing. 5. The gate must be located in an area not subjected to high stresses (the gate area could well be the weakest section on the part). 6. Plastic entering the part from the gate must impinge on a wall or mold member to create a small back pressure, avoid jetting. 7. The location has to be in an area that will minimize weld, flow or knit lines especially on class "A" surfaces. 8. The gate mark should not be located on an appearance or functional surface. 9. If the gate is of the type that must be trimmed, it must be located in an accessible area.

Methods of Venting
Injection molds must be vented to allow volatiles released from the plastic pellets during the plasitification process and the air trapped in the closed mold to escape. Adequate vents must be installed on the parting line, runner systems and in any place in the mold where entrapment of these gasses occur. The depth of the vent must be less than that which the plastic will flash. Experience has proven that the location of the vent in proximity to the gate will determine the depth to which plastic will flash. The closer the vent is to the gate the easier the material will flash. The longer the flow distance from the gate, the less likely plastic will flash at the same

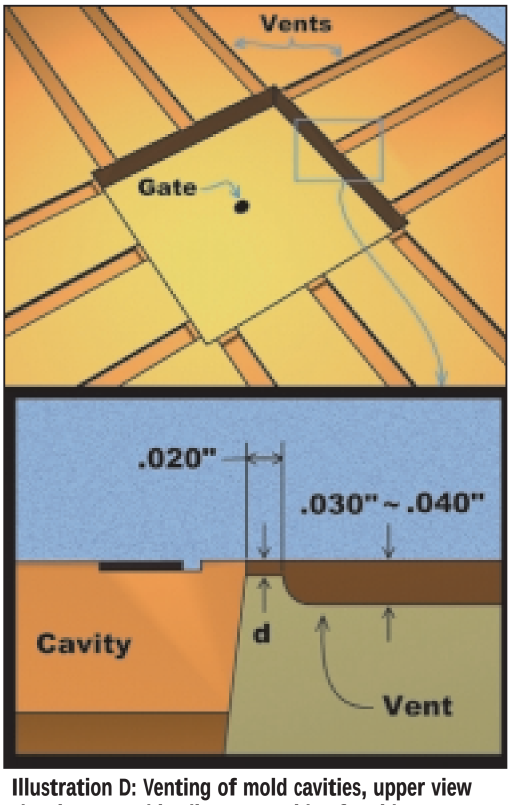

Illustration D: Venting of mold cavities, upper view showing vents bleeding to outside of mold. Lower view illustrates typical dimensions for vent and bleed off.

depth. Information available from the plastic material supplier is a very reliable source for technical information.

Vents
Vents (Illustration D) should be installed at the mold parting line. The depth should be slightly less then where flash will form. Vents should be 1/4 to 1/2 inch in width. After a short land length, typically .020 inch, the vent depth should be machined .03 to .04 deep and must lead to atmosphere outside the mold. Conventional runner systems should be vented also. Any gas that can be vented prior to arriving at the cavity is less volume that has to be allowed to escape. Typically runner system vents are twice the depth as the part if runner flash is not an issue and installed at the end of the runner and at junctions. Stopping short of the outside of the mold with any vent is extremely inefficient in allowing gasses to escape and considered bad practice.

Round parts with top gates, (Illustration E) lend themselves to relief around the entire part. A collector ring assists in collecting

MODERN PLASTICS, Issue date, Folio #, Date, Time
Version #
BLACK CYAN MAGENTA YELLOW
PMS ID
5% 25% 50% 75% 95%
RHP

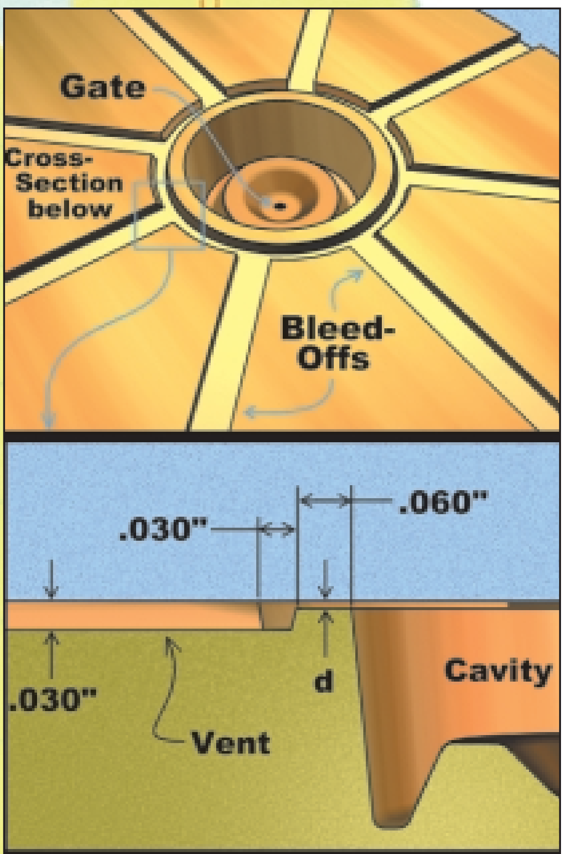

Illustration E: Top view shows perimeter venting of round cavity. Bottom view illustrates the vent and bleed off, typically used in relief venting.

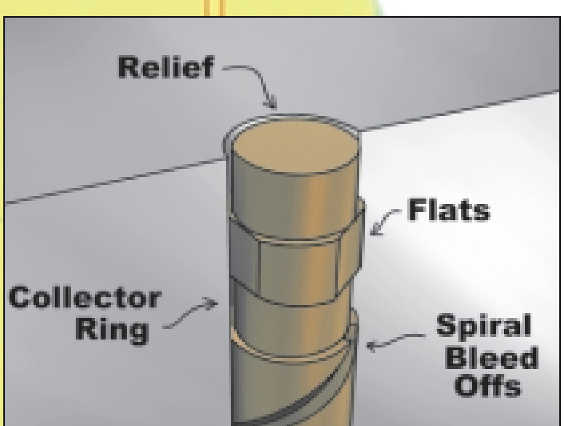

Illustration F: Method of venting at ejector or core pins.

the gasses and bleed offs allow them to escape to atmosphere. The volume of the bleed offs must equal or be greater than the volume of the collector rings to be efficient.

Areas in the cavity that can be vented other than the parting line are at insert lines, core pins and areas where sintered vents can be installed. Ejector pins are often overlooked as excellent places to install vents, as they are self cleaning due to their movement at ejection. An effective vent can be achieved by grinding relief on the top diameter (Illustration F) to allow the gasses to escape around the diameter to a collector ring and then spiral bleed offs to the ejector housing. This method keeps the ejector pin centered in the ejector pin hole, preventing shifting of the pin causing flash on one side of the pin and shutting off the vent on the other side and is much more effective than an undersize pin in an oversized ejector pin hole.

Mold Finish

The Society of Plastics Industry (SPI) developed and publishes a standard for mold finish that is universally used to specify the desired effect on the plastic part. The standard is based on four different methods, polishing with diamond, paper, stone or blasting. These categories then have three subgroups defining the last benching or polishing operation that will yield the desired finish or appearance on the plastic part. In addition to the listing of the SPI finish standard number and the corresponding operation, we have included the Ra and RMS measurement taken from a finished component. It is important to remember that the measurement is not part of the SPI standard, nor is it endorsed by SPI. It is simply included as a reference.

SPI Finish	Definition of Last Operation	Ra	RMS
A-13	Diamond Buff	0.6	0.8
A-26	Diamond Buff	0.8	1.0
A-315	Diamond Buff	0.9	1.2
B-1600	Grit Paper	2.1	2.6
B-2400	Grit Paper	3.8	4.9
B-3320	Grit Paper	4.8	6.2
C-1600	Stone	4.4	5.7
C-2400	Stone	8.0	10.2
C-3320	Stone	8.9	11.4
D-1	Dry Blast # 11	8.8	11.1
D-2	Dry Blast # 240 Al Oxide	13.2	17.0
D-3	Dry Blast # 24 Al Oxide	80.8	104.5

Acknowledgements
The injection mold design guidelines were written by Dr. Paul Engelmann, Associate Professor, Western Michigan University and Bob Dealey, Dealey's Mold Engineering, with the support of Dr. Dale Peters, for the Mold Marketing Task Group of the Copper Development Association. Kurt Hayden, graduate research assistant, WMU, generated the illustrations. Research conducted by WMU plastic program undergraduate students.

Disclaimer

These guidelines are a result of research at WMU and industry experience gained with the use of copper alloys in injection molding. While the information contained is deemed reliable, due to the wide variety of plastics materials, mold designs and possible molding applications available, no warranties are expressed or implied in the application of these guidelines.

Contact Information

Information on copper alloys is available from the Copper Development Association, 1-800-232-3282.

For more information about the use of copper alloys in tooling, please write in 687 on the reader service card.