PLASTICS ENGINEERING COMPANY
SHEBOYGAN, WISCONSIN 53082-0758 U.S.A

3518 LAKESHORE ROAD
POST OFFICE BOX 758
PHONE 920 - 458 - 2121
F A X 920 - 458 - 1923

Thermoset Transfer Mold Design Tips

When designing a mold for a transfer molded part, it is important to keep in mind that the goal is produce parts with the best quality, in as short a cycle as possible, with a minimum of scrap. To achieve this goal, you will need a mold that has a uniform mold temperature, has balanced fill, and is properly vented.

MOLD HEATING

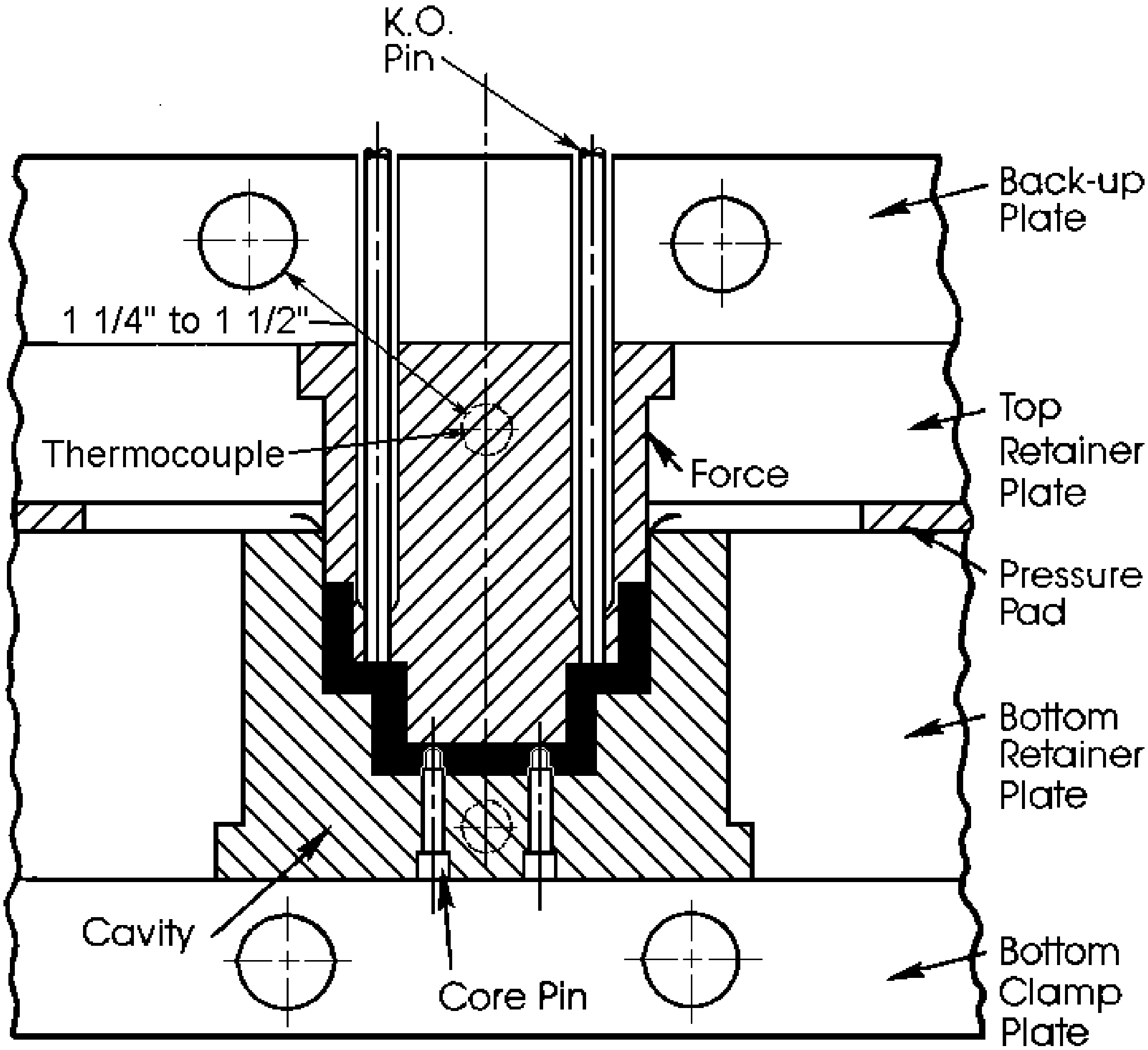
A **uniform mold temperature** means that the temperature of each half of the mold is the same (within ±3°C (5°F)) for all locations when the mold is heated by oil or steam. Molds that are heated with electric cartridge heaters can vary by as much as 6°C (10°F). A mold with a uniform temperature will fill easier and produce parts with less warpage, improved dimensional stability and a uniform surface appearance. Achieving a uniform mold temperature is dependent on your method of mold heating.

A mold that is **heated by steam** or **oil** will have a uniform mold temperature because the heat source maintains a constant temperature. However, oil, as a heat source, is only about half as efficient as steam. Therefore, when using oil to heat a mold, it is necessary to set the oil temperature higher than the desired mold temperature.

Electrically heated molds are more difficult to maintain at a uniform temperature because the cartridge heaters are constantly cycling on and off. When they are on, they generate a great deal of heat at the source but this heat must be distributed throughout the mold in a way that produces a uniform mold temperature.

To determine the amount of wattage needed to heat a mold, the use of the following formula might be helpful**: 1¼ kilowatts for every 45kg (100 pounds) of mold steel.** Note: This formula normally will allow the mold to be heated to molding temperatures in 1 to 2 hours.

Locating a heater on the centerline of the mold is not recommended, because the center of the mold is normally hot enough without adding any additional heat. Typically, the **cartridge heaters** are located in the support plates, with a distance of 65 mm (2½") between heaters. **NOTE:** Deep draw molds may need to also have heaters in the retainer plate. There should be a minimum of one **thermocouple** to control each half of the mold. In larger molds, it is recommended to have more than one thermocouple in each mold half. This will result in better control and more uniform mold temperatures. The **thermocouples** should be located in the "A" and "B" plates, between two heaters if possible and at a distance of 32 mm - 38 mm (1¼" - 1½") from the closest cartridge heater. This distance is to be measured from the edge of thermocouple hole to the edge of the cartridge heater hole. The distance from the thermocouple to the heater is important because a heater that is too close will cause the thermocouple to turn off the heat before the mold is at temperature. A heater that is too far away from the thermocouple will result in a mold that overheats and then gets too cool. Likewise, it is not a good practice to position a thermocouple so it senses the external surface temperature of the mold. If possible, it should be located 38 mm - 51 mm (1½" - 2") inside the mold, since the temperature taken there, is less susceptible to outside influences and therefore more stable.



BALANCE MOLD FILL

When transfer molding multiple cavity molds, it is important that all the cavities are filled simultaneously. The most common method to achieve a **balanced fill** is to make the runner length from the transfer pot to each cavity equal. This approach will work as long as the material flows directly from the transfer pot to the gate of the part. However, if the runner is divided two or three times between the pot and the gate, it is unlikely that the fill will be balanced. An effective way of **balancing the fill** is to have one main runner that extends from the last cavity on one end of the mold to the last cavity on the opposite end, with sub-runners feeding the individual cavities. To **balance the fill** of the cavities, flow restricter pins are placed in the subrunners. These pins are adjusted to inhibit the flow of material to the individual cavities so all the cavities are filled at the same time.

VENTING

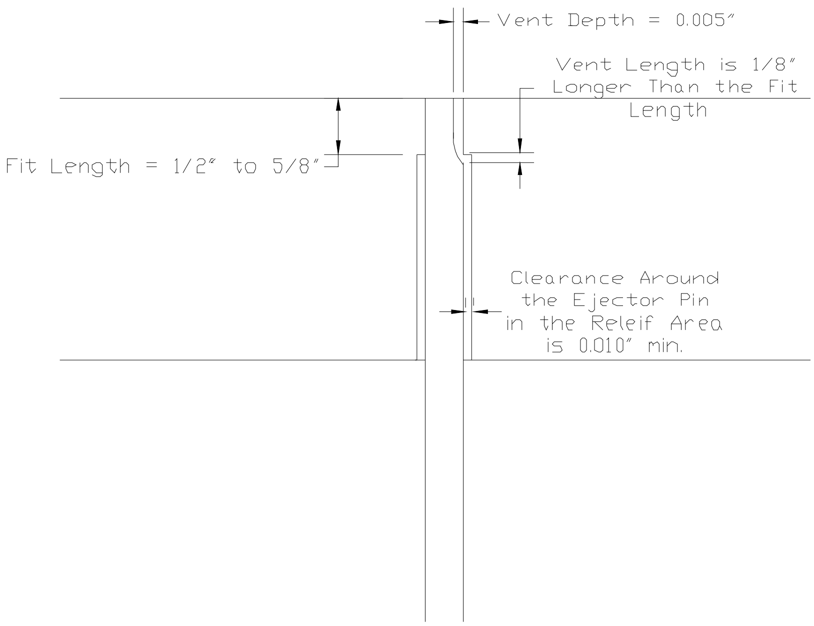
When molding thermosets, the polymerization process that takes place produces volatiles, which along with the air already within the cavity chamber can become trapped and superheat to 375°C - 425°C (700°Φ - 800°F). If the gases are not allowed to escape through vents, they may oxidize the lubricants leaving **burn marks** on the part. The vents allow the volatiles to escape to atmosphere. In addition to visual problems, improper venting will result in parts that cannot be filled, have dimensional problems, or have less than the expected physical and/or electrical strengths.

The first question that has to be addressed is **vent location**. It is important that all vents must lead to the atmosphere, otherwise the vent will be useless. Unless the part geometry shows some obvious locations for vents, a brief molding trial should be conducted to observe where the gas voids occur. Whenever possible, **vents** should be located in the moveable half of the mold, wherever a gas void or knitline is seen on a part.

Vents for phenolic parts should be 6 mm (¼") wide and 0.08 mm - 0.09 mm (0.003" - 0.0035") deep and **vents for polyester parts** should be 6 mm (¼") wide and 0.05 mm - 0.06 mm (0.002" - 0.0025") deep. The width is not as critical as the depth. A vent that is 0.025 mm (0.001") or less, is too shallow and may seal when the mold is closed. A vent that is 0.13 mm (0.005") is normally too deep and may not seal. As a result, internal cavity pressure will be low and the shrinkage, the physical and the electrical properties may not match data sheet values.

Of equal importance to the location and depth of the vents is the **vent length,** which is the distance from the part that the vent maintains its 0.08 mm (0.003") depth. The vent should be approximately 25 mm (1") long to allow pressure to build in the cavity after the material in the vent cures. After this point, the vent can be relieved to a depth of 0.25 mm - 0.50 mm (0.01" - 0.02"). To help the vent stay with the part, the corner of the vent at the part edge can be radiused or chamfered.

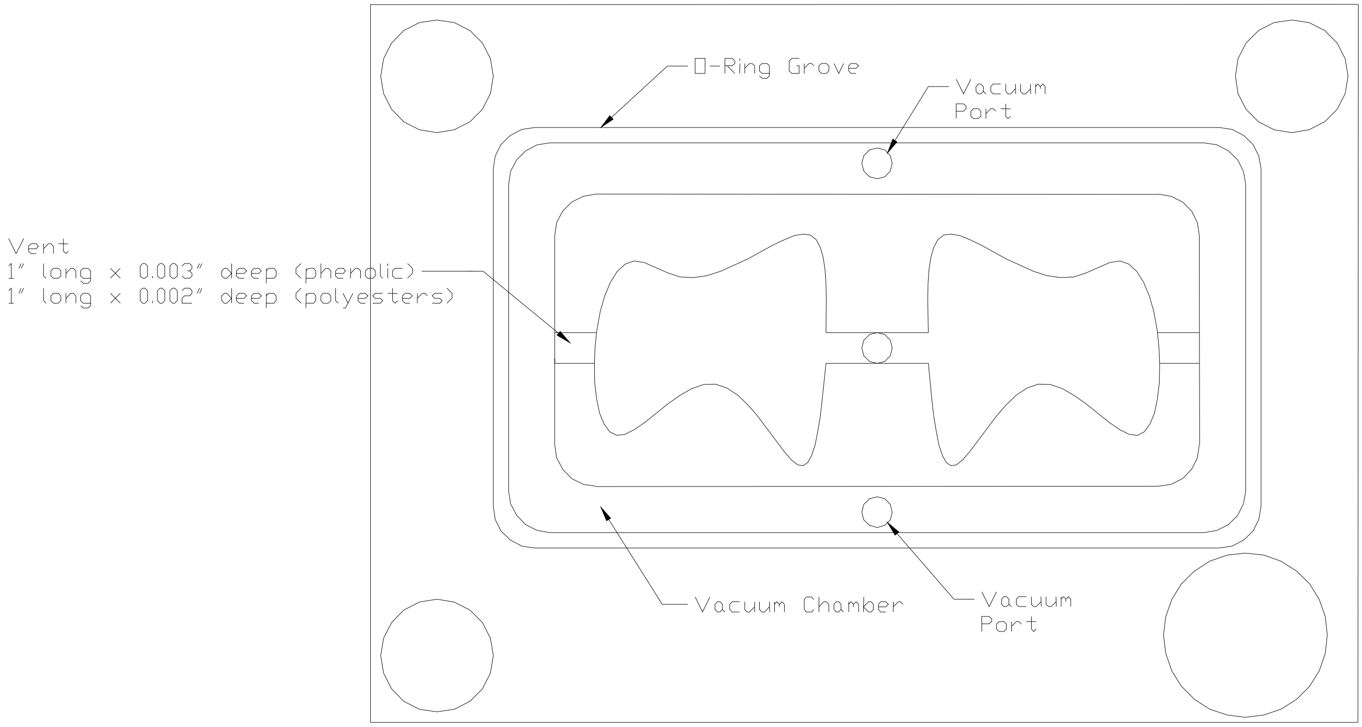
It is sometimes necessary to vent "dead" areas of the mold with **vented ejector pins.** Before adding the vents, an ejector pin should fit the hole in which it will operate within 0.025 mm (0.001"). A flat is then ground on the diameter no deeper than 0.13 mm (0.005") for a distance that will take the vent 3 mm (⅛") below the fit length of the pin. Normally, the fit length should be 13 mm - 16 mm (½" - ⅝") long. (See sketch below) In addition, the stroke of the ejectors should be long enough for the entire vent plus 3 mm (⅛") to come up above the bottom of the cavity. This is so the vent can be self -cleaning or so an operator can blow the flash off the pins.



Something that is often overlooked in venting is the polish. It is recommended that all vents be **draw polished** in the direction of flow to at **least** the same finish as the cavities and cores. They should be polished for their entire length including the relieved distance. If a mold is to be chrome plated, all the **molding surfaces** should be polished and plated including the vents.

Vacuum Venting

Some part designs are difficult to vent because of "dead pockets" or for other reasons. Also, some materials, such as thermoset polyesters, are difficult to adequately vent using conventional venting methods. In these situations vacuum venting is a good option to consider.



In a vacuum vented mold, the cavities are sealed inside of a vacuum chamber with an O-Ring. A vacuum of at least 21 in. of Hg is then drawn on the cavities. **NOTE:** A Venturi type vacuum pump will **NOT** be able to obtain this level of vacuum in the cavities.

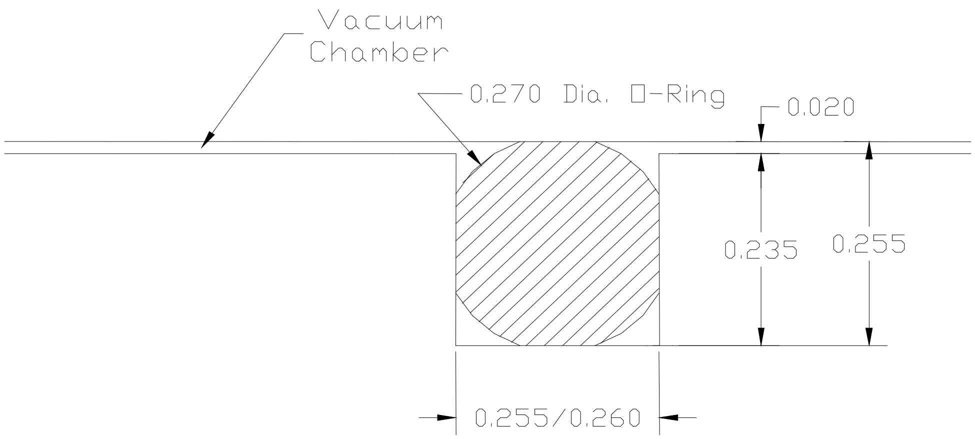
To check the amount of vacuum present in the mold cavities, we suggest closing the mold, inserting a plug in the transfer pot, putting a vacuum gage into the plug, activating the vacuum and then timing how long it takes to reach the maximum vacuum reading. This timer information is used to set the transfer delay so that once the vacuum is drawn, the molding compound can be transferred into the cavities. **NOTE:** Having an accumulator tank in the vacuum system will significantly decrease the amount of time needed to evacuate the cavities.

As can be seen in the sketch, the vacuum ports are located as far from the vents as possible. This is to prevent material from being drawn through the vents and plugging a vacuum port. The second vacuum port is a back up, in case the original port becomes blocked or plugged. **NOTE:** The vacuum system needs an inline filter between the mold and the vacuum pump, to trap any volatiles which would plug or damage the pump.

An O-Ring material that we have used successfully is high temperature silicone rubber that has a 60 to 70 durometer. One source of this material is McMaster Carr. Another source is Apex Molded Products Company, 3574 Ruth St., Philadelphia, PA 19134-2094 and their telephone number is (215) 289-4400 or (800) 221-8921.

A sketch for an O – ring groove is shown below and is designed to hold the O – ring in place and not have it pulling out of the mold with each shot.

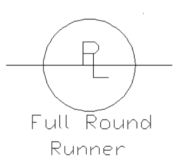
NOTE: The diameter shown in the sketch below is 0.270". However, other diameters can be used, as long as the proportions of the channel dimensions to the O-Ring dimension are maintained.



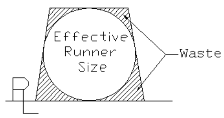
If you have any questions about the design of the groove or how vacuum venting can be incorporated into an existing mold, please contact the Technical Service Group of Plastics Engineering Company.

ADDITIONAL MOLD DESIGN TIPS

Runner Design - When designing runners for molds, there are a number of possible approaches. These include the standard **full round** with a centerline.

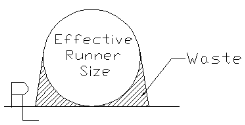


This is the most efficient runner, but in some cases it is necessary for the runner to be in only one half of the mold. See the figure to the left.



Standard Trapezoid Runner

A standard **trapezoid runner** is often used in situations that require the runner to be only in one mold half. The effective runner size is shown in the figure to the left. The four corners become "dead" areas with very little material movement. See the figure to the left.



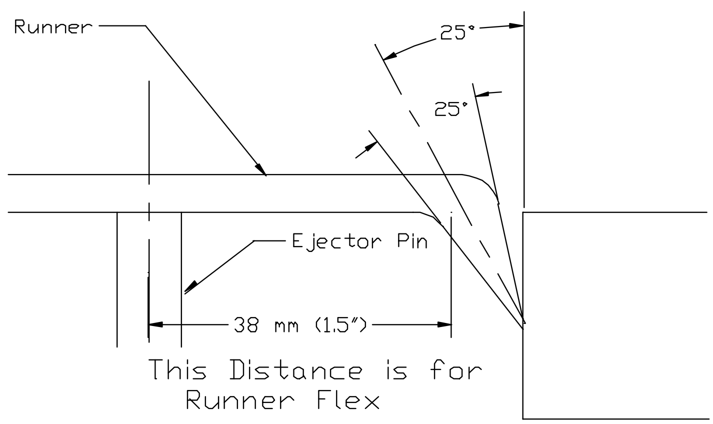
Modified Trapezoid Runner

To reduce the amount of scrap in the runner, a **modified trapezoid** runner design is suggested. This design reduces the dead areas without a significant change to the effectiveness of the runner. See the figure to the left.

Gates - The gates for thermoset molds are high wear areas of the mold and therefore, need to be designed with this thought in mind. The gate should be made using a replaceable insert so when the gate becomes badly worn it can easily be replaced. A gate should be made of materials that do not wear easily. Three materials commonly used for gate inserts are carbide, D-2 steel and CPM-10V particle steel made by Crucible Steel.

In addition to inserting the gate, we find that inserting the mold opposite the gate and at the impingement area in the cavity are beneficial. These areas are also high wear locations and will need some maintenance as the mold is run.

When designing an **edge gate** for thermoset materials, the width of the gate can be as small as 1.5 mm ($^{1}/_{16}$") but the depth of the gate should not be less than 1.3 mm (0.050"). A gate should be large enough to allow the part to fill without using excessive transfer pressures or requiring long transfer times. Transfer times of 3 - 8 seconds and transfer pressures of 7.6 MPa (1,100 psi) or less are desirable. Avoid using multiple gates on parts to minimize the number of knitlines. A **knitline** is created went two fronts of material meet. Knitlines are weaker than the rest of the part because there is not as much crosslinking that takes place across the knit as there is in the main body of the part. To keep the overall strength of the parts as high as possible, the number of knitlines should be kept to a minimum.



A second type of gate that is widely used in molds processing thermoset materials is the **subgate**. This type of gate is sometimes referred to as a **tunnel gate**. The advantage of a subgate is it shears off as the part is ejected from the mold. As a result, there is no need for a secondary operation to remove the gate nor is there any concern that the gate will project out from the part and become an assembly or a visual problem. In addition to the gate removal feature, the subgate can sometimes be designed to direct the flow of material towards a difficult to fill location. In this way, the part can be made easier to fill, which can have a positive effect on cycle times and scrap rates. Gate size is dependent on the size of the part. Typically 0.13mm (0.050") can be used for small parts and 0.20mm (0.080") for larger parts. There are some problems associated with using subgates that include:

- The tip of the gate breaking off and sticking in the mold. This is especially true for polyester molding materials and therefore, the use of subgates in molds for polyester parts is not recommended.

- The amount of steel at the parting line above the gate being too thin which results in the metal wearing away very quickly after the mold begins to produce parts.

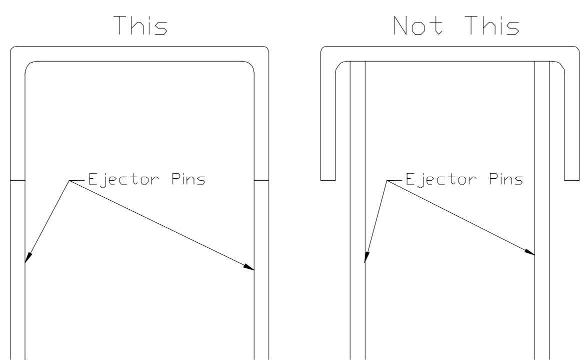
To reduce the likelihood of the gate tip breaking off and sticking in the mold, the tunnel needs to be well polished so all EDM pits are removed. Locating an ejector pin at least 38 mm (1½") from the tunnel allows the runner to flex and pull the gate out of the mold without breaking. It is also important to design the tunnel so the angle of incidence with the part allows the gate to pull, but keeps sufficient thickness of steel at the parting line to prevent damage. See the sketch for further clarification of how to design a subgate.

Recent developments in thermoset molding have shown that a part can be transfer molded with nearly all signs of the gate gone. This is done using a **gate cutter**. A gate cutter is a blade or a pin that is located in the mold directly below the gate. Immediately after the material is injected into the cavity, this blade is advanced forward to seal off the gate. Once the blade is in the forward position, the material cures against it, producing the same finish as the rest of the part. The only visible trace of a gate is a witness line.

Cavities and Core - In nearly all molds, the use of **inserted cavities** and **cores** is encouraged. The primary reason for this is in the event of an individual cavity or core being damage, that particular cavity can be removed from the mold and repaired while the rest of the mold is put back into service. Having individual cavities also allows for insert changes that make it possible to run multiple versions of the same basic part simultaneously. When the parts are very small and there is a large number of cavities, individual cavity inserts might not be feasible. In those situations, we would suggest using cavity inserts of 3 or 4 cavities. The materials most commonly used for cavity inserts are H-13 and S-7. Both of these materials will harden to Rockwell 52 to 54 Rc and can be polished to produce an excellent surface finish on the parts.

Ejector Pin Location and Design - Without ejector pins it is usually not possible to remove the molded part from the mold. The placement of the ejector pins is almost as critical as the location of the gate. The pins should push the part out of the mold without distorting it and without leaving an objectionable mark on the part. A secondary reason for having ejector pins is to aid in the venting of the mold.

Ejector pins should be located in the deepest points of the cavity or core. We specifically suggest that ejector pins be located on the deepest points of ribs and bosses. If ejector pins are not located correctly, the part has to be "pulled" out of the deeper areas or the mold. Parts that have to be "pulled" out of the mold are more likely to stick or be distorted during ejection. (See sketch below.)



Once the location of the ejector pins is determined, the pin size needs to be decided upon. Very small **diameter ejector pins** can be problematic because of their susceptibility to breaking. Therefore, ejector pins smaller than 2.4 mm ($^{3}/_{32}$") in diameter are not recommended. Another common problem is material flowing down around the ejector pin and jamming it so it breaks when the ejectors are actuated. To prevent this from happening, the hole for the pin should only be 0.025 mm (0.001") larger than the pin for a depth of 13 mm - 16 mm (½" - ⅝") from the cavity. Making it deeper can result in the pin binding and breaking.

To ensure that the ejector plate moves along the centerline of the ejector pins, it is suggested that the mold be equipped with a guided ejector system. In addition to aligning the ejectors, the guided ejector system moves the load of the ejector plate and the retainer plate from the ejector pins to the guide pins and bushings of the ejector system. While aligning the ejector holes in the mold with those in the retainer plate is always important, with a guided ejector system the alignment is even more critical.

While it is desirable to have the ejector pins located on flat surfaces, this is not always possible. It is sometimes necessary to locate ejector pins on contoured surfaces. Ejector pins located on contoured surfaces should be made to match the contour of the cavity. It will be necessary to key these pins so they will maintain their alignment with the contour of the cavity.

Polishing and Plating - The trend has been to cut back on **polishing** because of its high cost. Molds are being made that still show cutter marks on the non-visible areas of the parts. While this practice may save money in the construction of the mold, it may increase part costs due to high scrap and down time. The **non-polished areas** will generate frictional heat in the material as it passes over these areas. This added heat can cause the material to cure prior to filling the part. These unpolished areas can change the filling pattern of the material, which can result in gas being trapped in locations that can not be vented. For these reasons, it is suggested that all molding surfaces be polished to a minimum of SPI #2 rating. The **mold surfaces to be polished include** the cavities and cores, the vents, the gates, the runners, the transfer pot and the entire parting line. The reason for polishing the parting line is to insure that any flash that may occur on it will come off of the mold with a minimal amount of effort. When polishing a mold, care should be taken to be sure to always polish in the direction of draw. Vents need to be polished in the direction of material flow and they should have the same degree of polish as the cavity and core. Flat surfaces that have no influence on the part removal can be polished in any direction. When polishing deep ribs that were cut using the EDM process, it is important to be sure to polish out all of the EDM pit marks. Otherwise, there may be a problem with the rib breaking off of the part and sticking in the mold.

After the mold is completely polished, it is ready to be **plated**. Please keep in mind that any defect in the steel surface will not be covered by the plating, but will be accentuated by it. While there are a number of different types of plating available, to date, **chrome plated molds** provide the best part release with the best part finish. Because some materials have fillers that are incompatible with nickel, the use of nickel or electroless nickel to plate molding surfaces is discouraged. In addition, nickel plating lack the wear resistance of chrome plating.

The **surfaces to be plated should include** the cores, the cavities, the core pins, the ends of the ejector pins, the runner blocks, the vents, and the entire parting line. To protect the molding surfaces and to insure good part release, it is necessary to plate all the surfaces that were polished. After the mold is plated, it will be necessary to **repolish** the chrome because unpolished chrome plating may cause sticking.

Center Supports - Often we find that molds built to run thermoset materials have little or no support in the middle. This will result is heavy flash around the sprue and parts that vary in thickness from the sprue side to the side opposite. To resolve this problem we suggest installing substantial support pillars down the center of the mold between the parallels (50.8mm (2") dia. if possible).

High Centering the Mold - Sometimes the center of a mold will have heavy flash even with good center support. In these cases it may be necessary to do what we call "Doming the Mold" or "High Centering the Mold". This is accomplished by placing a 0.0508mm – 0.0762mm

(0.002” or 0.003”) shim on the support pillars in the center of the mold, which will cause the moving side of the mold to be slightly domed.

Side Locks - Any molds where maintaining the alignment of the mold halves is critical to meeting the quality requirements require non tapered side locks. They should be located on all four sides of the mold. The overall design of Progressive Components’ side locks is very good, since they have a longer engagement and are thicker.

Date Printed: January 29, 2009
Date Revised: January 7, 2009
Supersedes Revision Dated: October 9, 2008