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, and with minimum amount of scrap. To achieve this goal, you will need a mold that has a uniform mold temperature, balanced fill, is properly vented and is designed to dimensionally compensate for material shrinkage in all axis.

Mold Heating

A **uniform mold temperature** means that the temperature of each half of the mold is the same within 5°F (±3°C) 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 10°F (6°C). A mold with a uniform temperature will fill easier and produce parts with less warp, improved dimensional stability and a uniform surface appearance. Achieving a uniform mold temperature is dependent on your method of mold heating and tool design.

A mold that is **heated by steam** or **oil** will have the best 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. Heat is generated when the cartridge heaters turn on, but this heat must migrate throughout the mold in a way that produces a uniform mold temperature. Placement of the heating rods is important to achieving a balanced mold heat.

To determine the amount of wattage needed to heat a mold, the use of the following formula might be helpful**: 1.25 - 1.5 kilowatts for every 100 pounds (45kg) of mold steel.**

Typically, the **cartridge heaters** are in the support plates, with 2½" (65 mm) of space between the heater cartridges. **NOTE:** Deep draw molds may need to also have heaters in the retainer plate.

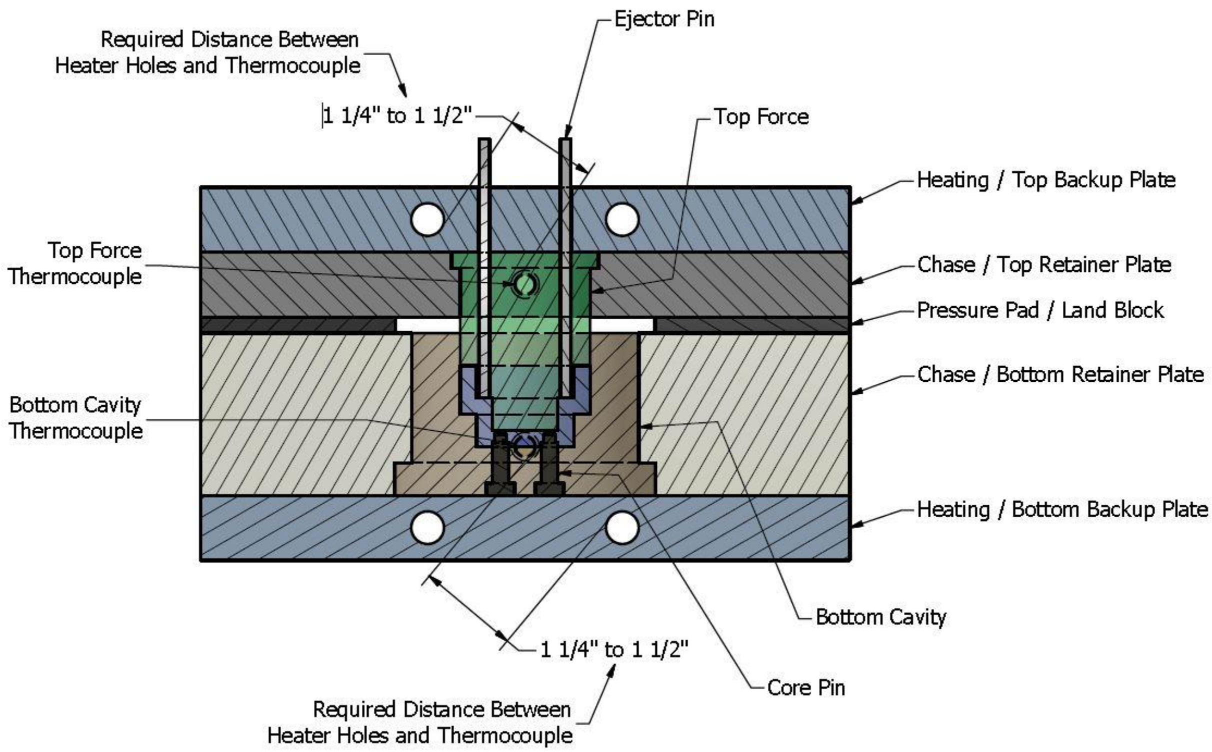
Caution should be taken to avoid locating cartridge heaters in deep draw areas of a mold due to the possibility of high mold temperatures in the proximity of a cartridge heater.

We suggest that 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.

How does a thermocouple work?

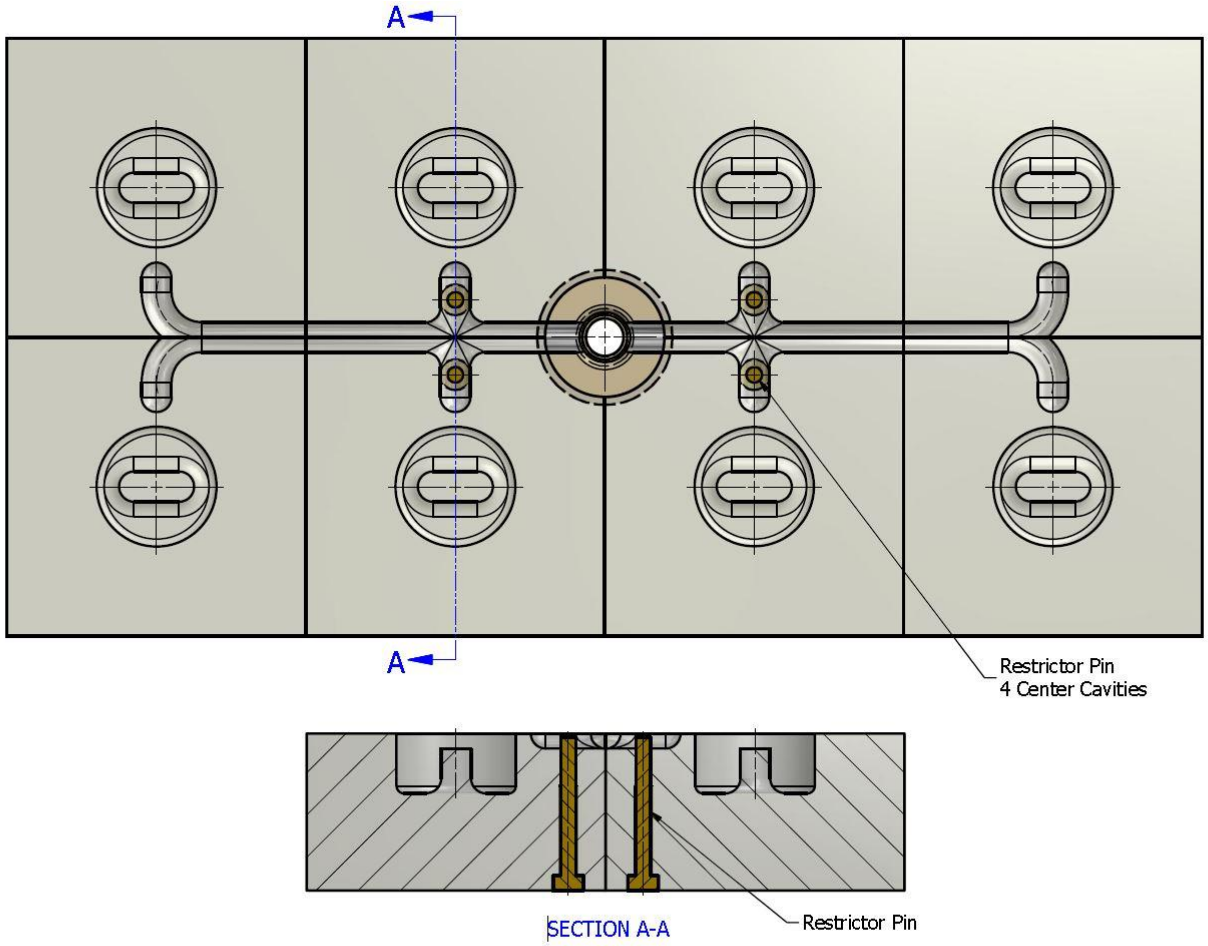
"A thermocouple is a sensor that measures temperature. It consists of two different types of metals, joined together at one end. When the junction of the two metals is heated or cooled, a voltage is created that can be correlated back to the temperature".

The **thermocouples** should be in the "A" and "B" plates, between two heaters if possible and at a distance of 1¼" - 1½" (32 mm - 38 mm) 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 cold. Likewise, it is not a good practice to position a thermocouple, so it senses the external surface temperature of the mold. If possible, the depth of the thermocouple should be as close to the cavity block as possible. Within ⅛" (3 mm) to touching the cavity block.



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 distance the material travels from the transfer pot to each cavity the same. This approach will work if 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 transfer 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 restrictor pins (also called balance pins) are placed in the sub runners typically in the interior cavities. These pins are adjusted to inhibit the flow of material to the individual cavities, so all the cavities are filled at the same time. Ejector pins within the runner system are good candidates for restrictor pins.



Venting

When molding thermosets, the polymerization process produces volatiles. The volatiles [3]700ºF - 800ºF (375ºC - 425ºC). If the gases cannot to escape through vents, they may oxidize the lubricants leaving **burn marks** on the part and staining on the mold surface. The vents allow the volatiles to escape to atmosphere. In addition to visual problems, improper venting will result in parts that are short filled, have dimensional problems, or have less than the expected physical and/or electrical properties.

It is important that the vent(s) lead to atmosphere otherwise they will be useless. Unless the part geometry shows some obvious locations for vents, a brief molding trial should be conducted to observe where the knit lines and/or gas voids occur. Whenever possible, **vents** should be machined on the moveable half of the mold, wherever a gas void or a knit line is visible on a part. We recommend locating vents in a manner that they do not flow into each other.

Typical vent designs are as follows. Please note that the width of the vent is not as critical as the depth. This is because a vent that is too shallow may seal shut when the mold is under tonnage and a vent that is too deep may not freeze off the material flow, resulting in lower physical and electrical properties than would be expected.

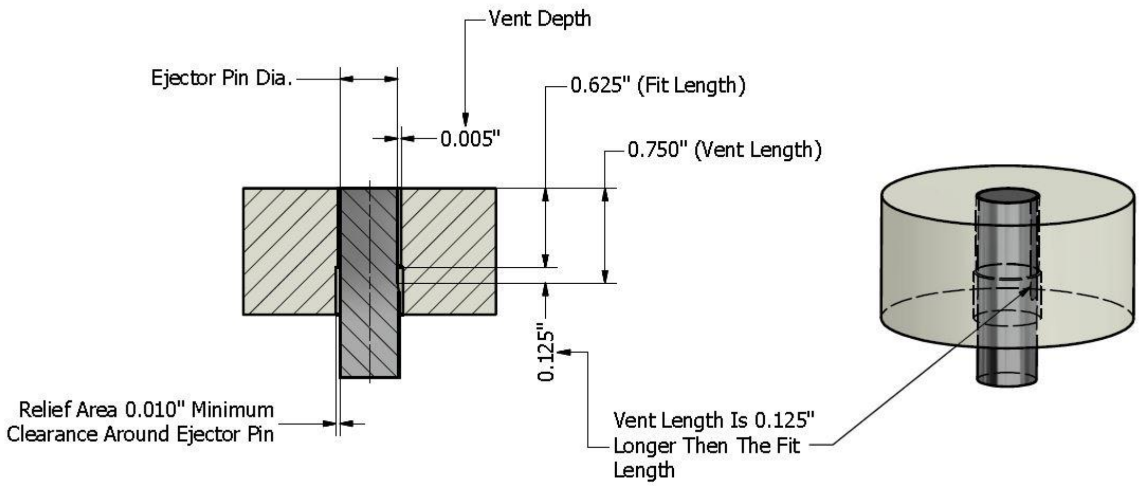
- **Vents for Melamine-Phenolic and Phenolic parts** should be $^3/_{16}$" – ¼" (5-6 mm) wide with a recommended depth of 0.003"- 0.0035" (0.08 mm - 0.09 mm).

- **Vents for polyester parts** should be $^3/_{16}$" – ¼" (5-6 mm) wide and 0.002" - 0.0025" (0.05 mm - 0.06 mm) deep.

The **vent length** for the above materials, which is the distance from the edge of the part to the vent relief to atmosphere, should be approximately 1" (25mm) long to allow pressure to build in the cavity after the material in the vent cures (freezes off). After this point, the vent can be relieved to a depth of 0.01" - 0.02" (0.25 mm - 0.50 mm). **Breaking the edge**, which is chamfering or adding a radius to the corner of the vent at the part edge will help the vent stay with the part during ejection.

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

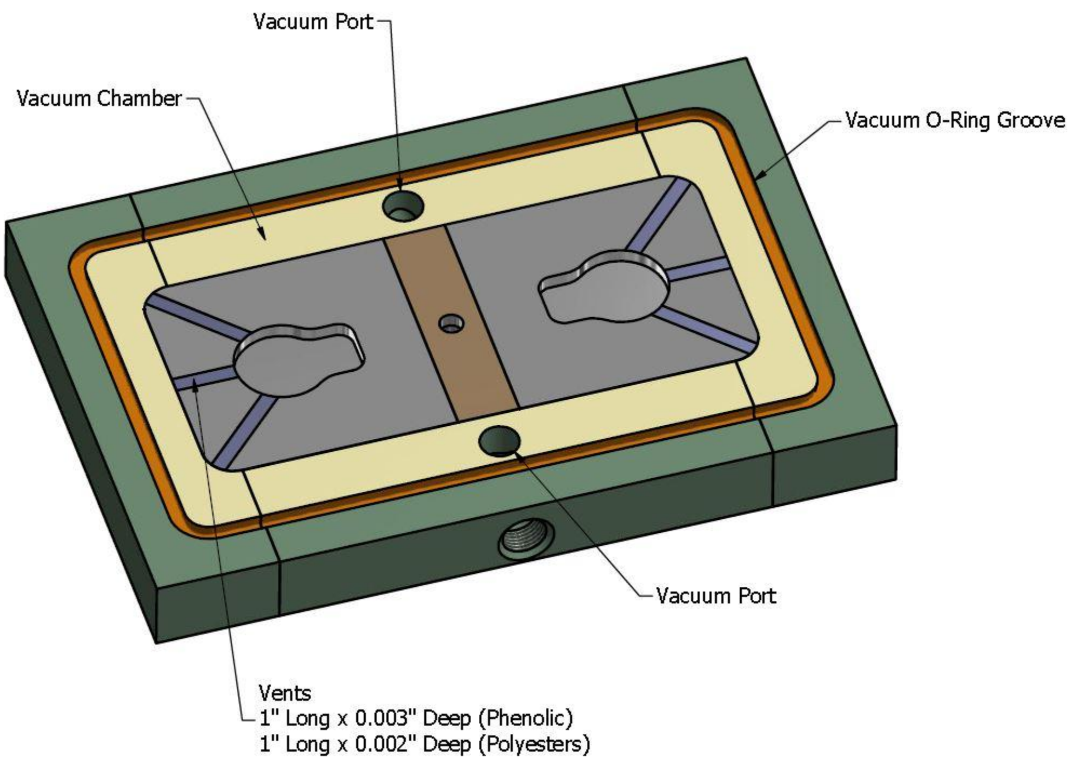
Vented Ejector Pin



Something that is often overlooked in venting is the polish. The lack of polishing the vent will cause it to stick to the tool. 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. However, vacuum venting a tool for molding phenolic materials is not as effective as it is for polyester material, and it should be considered as a last resort for phenolic tools.



In a vacuum vented mold, the cavities are located within a vacuum chamber which is sealed by an O-Ring. A vacuum of at least 21 in 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.

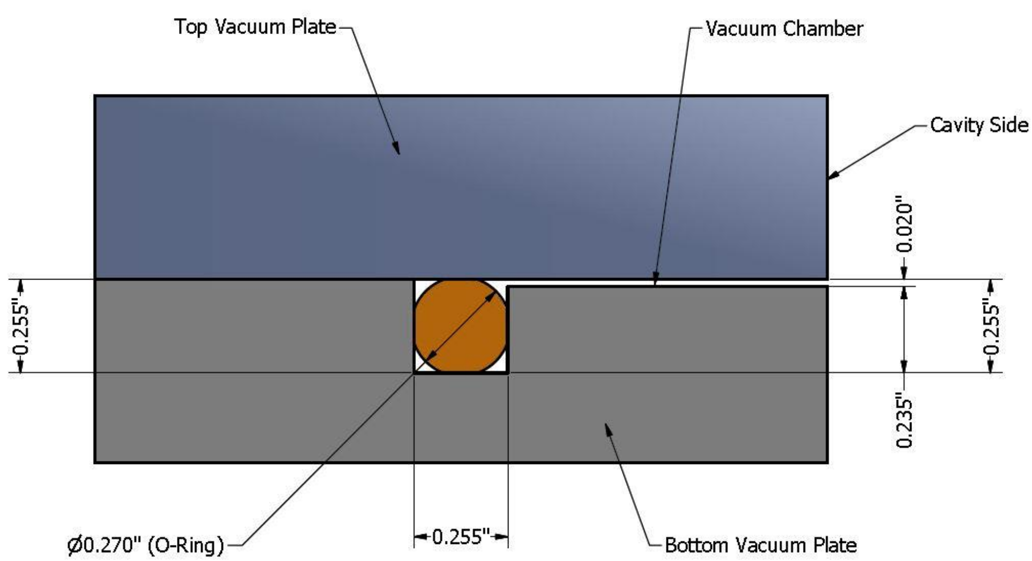
To check the amount of vacuum present in the mold cavities, we suggest disconnecting one of the vacuum lines and plugging it on the vacuum pump side, closing the mold, and inserting the vacuum gage in the open vacuum orifice, activating the vacuum, and

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 above drawing, 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 backup 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 (Viton) that has a 60 to 70 Durometer hardness. One source of this material is McMaster Carr.

A drawing 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 drawing below is 0.270" (6.9mm). However, other diameters can be used, if the proportions of the channel dimensions to the O-Ring dimension are maintained.

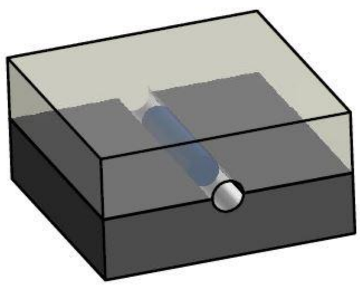


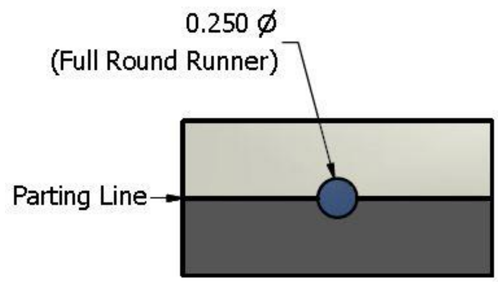
Additional Mold Design tips

Runner Design

When designing runners for molds, there are several 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.

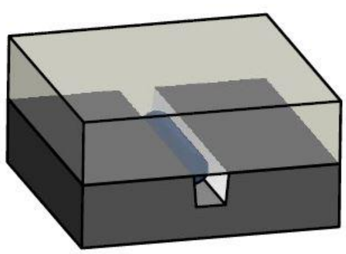
Full Round Runner

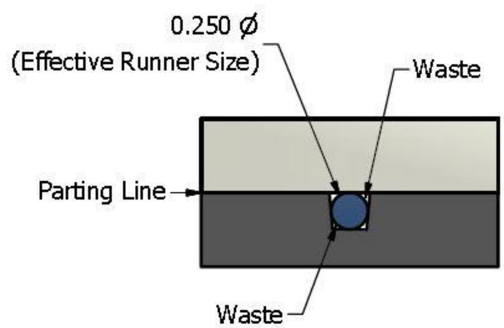




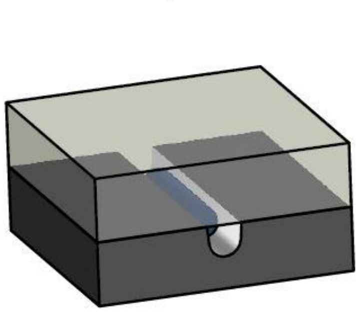
A standard **trapezoid runner** is often used in situations that require the runner to be only in one mold half. The effective runner size is shown in the figure to the left. The four corners become "dead" areas with nearly no material movement.

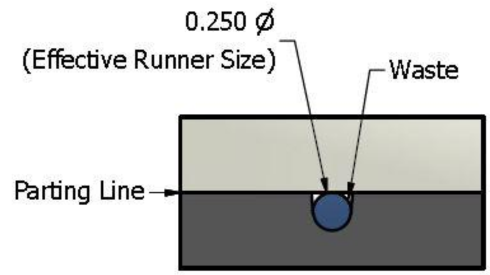
Standard Trapezoid Runner





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.





Gates

Gates - The gates for thermoset molds are high wear areas of the mold. The gate should be made using a replaceable insert so when the gate becomes worn it can easily be replaced. A gate should be made of materials that do not wear easily. Materials commonly used for gate inserts are D-2, M-2 and Ferrotic steel.

In addition to inserting the gate, it is beneficial to insert the area opposite the gate. These areas may be 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 $^{1}/_{16}$" (1.5 mm) but the depth of the gate should not be less than 0.050" (1.3 mm). 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 1,100 psi (7.6 MPa) or less are desirable. Avoid using multiple gates on parts to minimize the number of knit lines. A **knit line** is created when two flow fronts of material meet. Knit lines are weaker than the rest of the part because there is less crosslinking that takes place across the knit than in the main body of the part. To keep the overall strength of the parts as high as possible, the number of knit lines should be kept to a minimum.

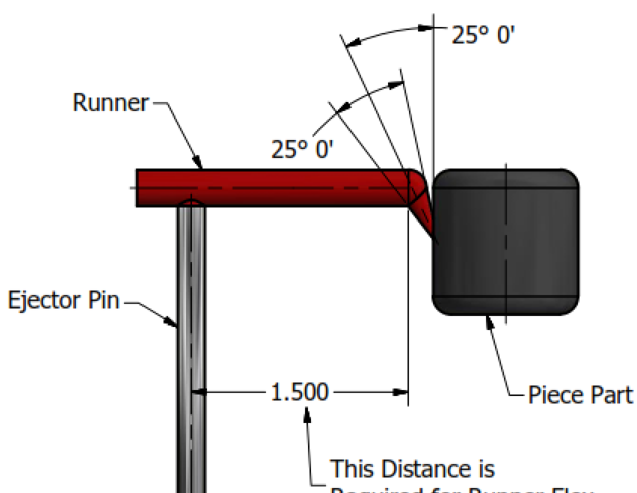
A second type of gate that is widely used in molds processing thermoset materials is the **sub gate**. This type of gate is sometimes referred to as a **tunnel gate**. The advantage of a sub gate is it shears off as the parts are ejected from the mold. As a result, there is no need for a secondary operation to remove the gate vestige, 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 sub gate 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, 0.050" (0.13mm) can be used for small parts and .080" (0.20mm) for larger parts. There are some problems associated with using sub gates 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 sub gates 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 1½" (38 mm) 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 drawing on the next page for further clarification.

Subgate

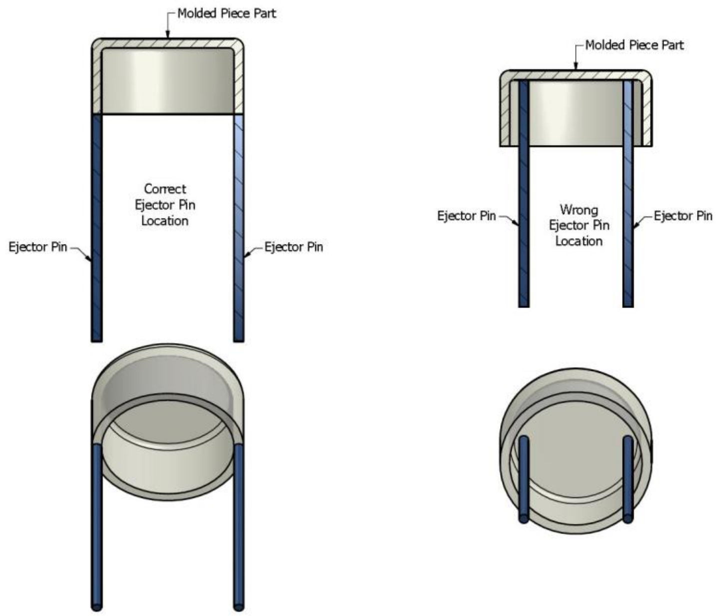


A **gate cutter** can be used to mold a part with nearly all signs of the gate gone. When transfer molding, gate cutters are typically not used. A gate cutter is a blade or a pin that is in the mold directly below the gate. Immediately after the material is injected into the cavity the blade is advanced 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. This process requires a modified press control system for the gate cutter sequence.

Cavities and Core - In nearly all molds, the use of **inserted cavities** and **cores** is recommended. The primary reason for this is in the event of an individual cavity or core being damaged, 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 makes it possible to run multiple versions of the same basic part simultaneously. When the parts are very small and there are multiple cavities, individual cavity inserts might not be feasible. In those situations, we suggest using cavity blocks consisting of 3 or 4 cavities. Common steel grades used for cavity inserts are H-13 and S-7. Both 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 typically not possible to remove the molded part from the mold. The ejector pins should push the part out of the mold without distorting it and without leaving an objectionable mark on the part.

Ejector pins should be 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 must be "pulled" out of the deeper areas of the mold. Parts that must be "pulled" out of the mold are more likely to stick or be distorted during ejection. (See drawing below.)



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

To ensure that the ejector plate moves along the centerline of the ejector pins 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 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

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 cannot 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 ensure that any flash that may occur on it, will come release from 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 EDM pit marks. It is necessary to polish all surfaces thoroughly to minimize the potential for part/flash sticking.

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 several 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 electro less nickel to plate molding surfaces is discouraged. In addition, nickel plating lacks the wear resistance of chrome plating.

The **surfaces to be plated should include** the cores, the cavities, the core pins, and the ends of the ejector pins, the runner blocks, the vents, and the entire parting line. To protect the molding surfaces and to ensure good part release, it is necessary to plate all the surfaces that were polished. After the mold is plated, it will be necessary to **re-polish** 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 stationary (sprue) side to the movable side. To resolve this problem, we recommend installing substantial support pillars down the center of the mold between the parallels 2” (50.8mm) diameter if possible.

Side Locks

We suggest using side locks for all injection molds to maintain the best mold alignment.

Printed: April 28, 2026
Revised: April 28, 2026
Supersedes: October 6, 2015