Content

Background P2~3

Part A: Over-molding (Insert-molding) P4

1 Machine Design P4~5
2 Transfer Mechanism P6
3 Product Design P7~9
4 Mold Design P10~12
5 Typical Applications P13~16

Part B: 2-k mold / Multi-shots molding P17

a Machine Design P17~20
b Transfer Mechanism P21~23
c Product Design and Mold Design P24~28
d Typical Applications P28~35
Background

The presence of an already molded component during the second and subsequent molding stages makes **multi-material injection molding different from traditional injection molding process**. Therefore, designing multi-material molded objects requires addressing many additional manufacturability considerations. We expect that the manufacturability analysis techniques presented in this paper will help in decreasing the product development and mold design issues for the injection mold multi-material objects.

Over the past few years, a wide variety of multi-material injection molding processes have emerged for making multi-material objects, which refer to the class of objects in which different portions are made of different materials. Due to fabrication and assembly steps being performed inside the molds, molded multi-material objects allow significant reduction in assembly operations and production cycle times. Furthermore, the product quality can be improved, and the possibility of manufacturing defects, and total manufacturing costs can be reduced.

There are fundamentally three different types of multi-material molding processes. In multi-material injection molding, multiple different materials are injected into a multi-stage mold. The sections of the mold that are not to be filled during a molding stage are temporally blocked. After the first injected material sets, then one or more clocked portions of the mold are opened and the next material is injected. This process continues until the required multi-material part is completed. Multi-stage injection molding is perhaps the simplest form of multi-material molding. It involves either simultaneous or sequential injection of two different materials through either the same or different gate locations in a single mold. Multi-shot injection molding is the most complex and versatile of the multi-material molding processes. It involves injection the different materials into the mold in a specified sequence, where the mold cavity geometry may partially or completely change between sequences. Over-molding (Insert-molding) simply involves molding a resin around a previously-made part. Each of the three classes of multi-material molding is considerably different. Each specific multi-material molding process requires its own set of specialized equipment; however, there are certain equipment requirements that are generally the same for all types of multi-material molding. Techniques described in this guild line are applicable to over-molding and multi-shot molding.
1 Machine Design.

Different plastic materials are injected at different stages of the process using different cavity geometry. So a plastic part/insert is first molded/made and then transferred to a different cavity, to be over-molded by the second material/color by a normal injection machine or a multi-material injection machine.

Solution 1

Using two normal injection machines

Solution

Using only one multi-material injection machine, and part transfer manually or with robot from the first short to next.
2 Transfer Mechanism

There are two ways to transfer part, manually or with robot. The first shot must cold down fully if it is supposed to be transferred manually, otherwise the shrinkage will be not certain which can influence the part quality. If you choose the way with robot, the cycle time also must be certain as it also related to the shrinkage. The key is the fitting of the second shot for both ways, it is recommended to get the first shot ready and do the final machining and fitting of second shot.
3 Product Design

- The wall thickness of the substrate and over molding should be as uniform as possible.

- Transitions between wall thickness should be gradual to reduce flow problems, such as back fills and gas traps.

- Deep, un-vent-able blind pockets or ribs should be avoided.

- Long draws should have a draft

- The thickness should be less than or equal the thickness of the substrate to prevent warpage, this is especially critical for long, flat part.

- The flow length is another consideration, it must be always within the maximal flow length, for safe reason 80% of maximal flow length is recommended. This could be verified by mold flow analysis always.

- In case there is more than one injection area for over molding, the filling must be balanced, via different gate size or even use sequential hot runner system with valve pins.

- Shut-off design for sealing of the second injection must be considered. There are two different types of shut-off designs with the purpose of minimizing flashing, minimizing press marking and undertake injection pressure of over molding part.

![Diagram: A shut-off design with an indentation can help ensure good adhesion.](image)
To reduce the opportunity for peeling, the over mold should be considered to design like below.

Material bonding must be always considered.
4 Mold Design
Mold design for an **over mold part** is not easy, there are a lot of points need be considered while designing the mold, some typical and common points are selected out and listed as blow, we must ensure that every point has been considered and sort out at design stage.

- Using certain features for orientation, ensure the first shot part is fixed in position for over molding.

- As the first shot part need be ejected out form mold so it must have shrinkage in some degree, the mount of shrinkage is difficult to estimate it more or less depends on part shape and cooling time, hereby we must study the part and define the injection process and cycles carefully, the best way is that finish the first shot and cold it down according to defined process and measure it, and then adjust the shrinkage of the second shot, so manufacture mold accordingly.

- The inject pressure will push the substrate intensively in case the substrate is not strong enough and well supported it perhaps will move during injection of the second shot. Hereby we must check the part design and fitting between substrate and mold to ensure the part is well supported at injection area of the second shot. In some cases we have to ask part design to redesign the part if there is not enough support.
It is often necessary to provide cosmetic “style” lines between the substrate and the over molded part. There needs to be a certain amount (around 0.05mm~0.2mm) of preload (interference) applied to the shutoff (boundary) areas on the substrate. This results in slight deformation of the area in contact. This aesthetic issue can be avoided by creating an intentional area such as the groove, for this to take place. Style grooves are also necessary when the part is textured. On the other hand the injection pressure exerted by the mold on the substrate, the textured areas will be compressed and a noticeable “shiny” area will develop. This can be eliminated by utilizing the groove as the “pinch” point.

We should always use mold-flow analysis to check air traps to ensure the venting must be sufficient for over molding, otherwise the part will be difficult to be filled up, or result in some issues such as flash and burn making.

The substrate doesn’t have to fit to mold completely except over molding area and sealing area, in order to save machining and fitting work, we should try to avoid unnecessary fitting surface as much as we can.

The injection gates often result in more or less defect such as flow marking and sink mark and so on, so it can’t be placed at/ against high demands surface for over mold, otherwise must get confirmation from customer.

Warpage /Deformation of substrate part can influence the final quality of over molding, big warpage /deformation should be avoided always, we should always use mold-flow analysis to understand the warpage /deformation and to determine if the part is suitable for over molding.

The sealing area is always tricky, the sealing surface must me uniform and smooth without parting line steps, so we should try to avoid split line on sealing area, that means try to avoid separate insert with split line at sealing area for both shots.

The molding process generates hundreds of thousands of pounds of pressure against the cavity walls of the mold. When those cavity walls are plastic, this can spell disaster for the substrate. Features under the 2nd shot in support areas need to have clearance built into the mold to accommodate for deviations in part tolerance. These features, such as ribbing, should be kept to a minimum in those areas. Second shot material below is first shot material, if the first shot have ribs, then the clearance can’t make too much, otherwise the second shot material will push the first shot material go down.

Any kind of sharp edges must be avoided to tough substrate part, sharp edge will leave press marking or even damage the part. All surfaces need be smooth and all edges need radius if they are going to touch the substrate part.
5 Typical Applications

Mold Number: ST 2013-022

Mold Number: ST2013-023
DONGGUAN SINCERE TECH CO.LTD
2-K mold, Multi-Component Injection Mold Design Guild Line

Mold Number: ST 2013-045

Mold Number: ST 2013-136 & 137
DONGGUAN SINCERE TECH CO., LTD
2-K mold, Multi-Component Injection Mold Design Guild Line

Mold Number: ST 2013-157&158

Mold Number: ST 2013-165

Mold Number: ST 2013-209&207

Address: 37#, JiangXia Lu, YuanJiang Yuan Cun, Changping Town, Dongguan City, Guangdong Province, China, 523000
Website: https://www.plasticmold.net/ Email: info@sinceretechs.com
a Machine Design

The size and tonnage of a machine will be dictated by the size of the tool. After a machine is selected, we must determine the best location for the second injection unit, which will help guide the machine configuration. Here are four of the most common configurations:

- **Parallel:** This method has side-by-side injection units located in holes that go through the platen. However, these two holes limit the machine from being used for anything other than multi-component molding.

Another solution when parallel injection is needed is to have one injection unit utilizing the central hole as in a traditional molding setup and running a small hot runner through the platen in a second, smaller hole, interfacing with the tool mounted on the other side. In this instance, the second injection unit is positioned at an angle that allows for the machine to be used for more than just parallel, two-component molding. Both configurations of parallel molding use relatively little floor space.
Perpendicular: This is probably the most common machine setup, and is middle-of-the-road in terms of cost. The second injection unit is located off of the back—non-operator—side of the machine. It takes up more floor space but allows for easy robotic removal of the part from the top.

Vertical: This method has a small footprint and is ideal when smaller injection units and shots are needed. It is also limiting in terms of robotic part removal, so a part-drop ejection is best.
- **Over/under:** This method is parallel injection with the second barrel injecting over the top of the fixed platen. Two major benefits of this configuration are that it’s compatible with top-entry robotics and that the main injection unit doesn’t have an altered location, meaning the machine can also function with a conventional single-material mold.

Overall, having the proper sequencing will lower cycle times and make the production process as efficient as possible. With a rotary table, where the cavities are independent, both materials can be injected at the same time. Both injection units will start together, but whichever unit has the longest process will determine the cycle time. If the cavities aren’t independent—as with a core back, for example—the sequence will need to be adjusted.
Most newer machines have control software that can handle the extra inputs and outputs associated with multi-material molding. But be aware that this is not always the case. Some older control systems or low-level controls on retro fitted machines might not be able to accommodate your requirements. Talk to machine supplier to ensure everything is up to par. Multi-material molding comes with a variety of challenges, but it’s a field almost any tool maker should be able to enter with a little bit of knowledge and network of support. There are many options and each application is unique. There’s no one best solution for all applications, so it’s important to work closely with customer, molder, machine manufacturer, and material supplier to ensure we have best setup for the part we’re trying produce.

**b Transfer Mechanism**

The more common—yet more complex—way of multi-material molding is to use just one machine with a rotary core. This means the machine must be large enough to accommodate the core and incorporate multiple injection units, but the approach still has a number of advantages over the two-machine method. There are a number of ways to accomplish rotary core over molding, and we’ll discuss three of them here: turning cores/plate, turntable (rotary platens) and core back.

Depending on the specifics of your application, each of these options has different benefits and carries greater or lower costs. Sorting through these circumstantial differences to determine the method that will maximize performance, quality, and cost-effectiveness is another reason it’s important for part and mold designers, processors, material suppliers, and machine providers to work together from the beginning of a project.

**TURNING CORES/PLATE**

Turning cores/plates are used when trying to mold the same material on both sides of a parting line and completely encapsulate a portion of a part, for example, surrounding the handle of a toothbrush with a second material.

Turning plates require the use of two different cavities and cores. This operation functions by molding the initial component, opening the mold, and moving the part into a second cavity by means of the turning plate moving on a horizontal axis. The mold closes. The machine injects the second material to create a finished part, while simultaneously injecting another initial component in the first cavity. The finished part is ejected from the second cavity. The turning table moves the new initial component into the second cavity, and the process repeats.

One major advantage of turning plates is that they offer quick cycle times, since the injection of both cavities is parallel. While turning plates usually produce good part quality, the technology limits the number of cavities, and therefore parts.
TURNTABLE (RO�ARY PLATENS)
A turntable is similar to a turning plate, except that instead of the plate turning, the B side mounting surface rotates the mold, usually 180°. This allows half of the mold surface (the A side) to be used for part production. The B side remains on the core and is not open for the second material.
This method requires a mold to be located directly in the center of the turntable, both vertically and horizontally. It will be important to incorporate a locating device, such as pins or keys, to ensure that the part is lined up correctly when the table rotates to fill the second cavity. The table can also be set up to accommodate more molds or to oscillate instead of completely rotating. Almost all turntables are servo driven, making them very precise and additionally helping with this critical centering action.
CORE BACK
The core back is used when you need to pull a core out of a mold before injecting the second material. The initial component is molded and cooled, the core is pulled back, the second material is injected into the volume exposed by the retraction of the core, the part is cooled and then ejected.
Core-back molds generally cost less and allow a moldmaker to incorporate a high number of cavities into a tool. This process does have longer cycle times, since it requires injection of one component after another and two separate cooling cycles.

Product Design and Mold Design
First of all, we must think about some basic points when we receive an inquiry of multi-material mold
Can this part design be two shot molded?
Are these two materials bonding well?
Can it be molded automatically?
Part design will determine how to make the tool design. In most situations we would like to be able to keep the first shot part on the core in the ejection half of the mold. If this can be done we know the part location is as accurate as the location of its core. Also, if the mold design requires rotation, the part is securely held in place during this process. To determine if this can be done we have to consider the two different portions of the part, their configurations and locations, and where the core side of the mold needs to be. Other major considerations at this time include how to get material to the individual shots and how to eject the parts and runners.

A simple and relatively inexpensive way to produce an automatic multi material mold is to utilize core pull (core back solution) and retract a portion of the core to make second shot cavity space available. This is generally less expensive because we only need one core and one cavity per part and can be much smaller mold size. This also in turn will generally allow for more cavities in less space. However, this can only be done if the bottom of the second shot area can have the same configuration or contour as the top of the retracted core.

Figures 1A and 1B and Figures 2A and 2B show cross sections of such parts. The second shot cavity space is made available by the core back solution.

In figures 3 below, we see a part that can not be two shot molded by core back, as designed.
A core back will not work without a part change. This part could be molded in a turn table or turn core. However, this type of construction is considerably more expensive. This is a situation where a slight modification could reduce mold cost by allowing a core back solution. See below illustration.

If the last shot contour could have a small flat of last shot material around, without compromising the parts function, a core back would work. The retractable core would seal against the flat and keep first shot material out of the last shot cavity.

If the part cannot be changed, turn table solution could be used. In this type of design, the first shot part would be molded on a common core in the first shot. Upon mold opening, this first shot part, on the core, would be rotated 180 degree to line up with a second shot cavity. There would be an identical core lined up with the now empty first shot cavity. When the mold is closed the second shot cavity accepts the first shot part and provides the open cavitation for the second shot material. See below illustration.
In this type of mold construction, for a one out tool there would be two identical cores, one first shot cavity and one last shot cavity. Basically this type of design will work whenever the main core side features can all be of first shot material or can readily be shut off on the first shot by steel from the cavity side.

Below illustration shows a common core with turn table applications.

Another common core with turn table application as blow, again the cores are identical. Or common can the cavities are different.

Another type of automatic multi material molding is to utilize different cores and cavities in both the first and second shot positions. To do this when there are core configurations that need to differ from the first shot to the second shot, is transferring the parts on a stripper plate or cores (Turn core solution). In this method the first shot part stays on the stripper plate after the mold opens and the stripper plate pushed these parts off the first shot cores. There has to be a feature on or in the stripper plate/core to securely hold these first shot parts while the core/plate is rotating. When the mold recloses the second shot cores push the part out of the features in the stripper plate and into the last shot cavities.
Gating options.

As a common question, how to get material to the individual shots is a major consideration. We have used every style of gate in multi material molds, at one time or another that we have used in single shot molds.

In many cases, and virtually all multi material molds, we want to get rid of the first shot gate and runner prior to molding the last shot. Where possible a tunnel gate is ideal. Quite often pin point gates, either hot runner are used on the first shot. This may be the only way to get the first shot cavity area. Gate vestige or appearance may be a concern in this application, and needs to be considered.

A tunnel gate may be necessary on either shot to get into the bottom/side of the cavity area. If tunnel gating is used on turn core/plate and table mold, an auxiliary ejector system will be required to eject the gate and runner.

Other possibilities would include tunnel gating into an ejector pin under the part of either shot of coring a flow channel through a first shot part to get last shot material to an inner feature.

d Typical Applications

TURN CORES/PLATE
TURN CORE-1

TURN CORE-2
Refer to attachment TRUN CORE for detail of design.

**TURN TABLE (ROTARY PLATENS)**
DONGGUAN SINCERE TECH CO.LTD
2-K mold, Multi-Component Injection Mold Design Guild Line

TURN TABLE-7

TURN TABLE-8
TURN TABLE-9

Refer to attachment TURN TABLE for detail of design.

CORE BACK

CORE BACK-1

Refer to attachment CORE BACK for detail of design.