injection WORLD

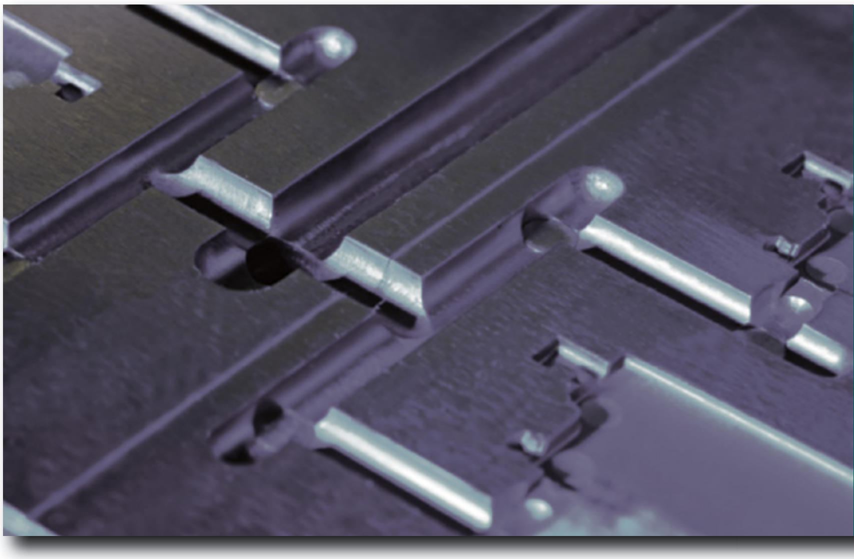
April 2013

POLYMERS FOR LED APPLICATIONS

AUTO PP COMES UP TO SCRATCH

BIOPLASTICS CAN STAND THE HEAT

NOVEL SOLUTION FOR MULTI-SHOT



In the second part in this three-part discussion on mould design, **André Eichhorn** provides some advice on avoiding problems due to poor venting and cooling of the cavity

Mould venting and cooling

As stated in the first part of this discussion on effective and structured mould design, venting of the cavity is very often overlooked at the design stage and only given any real consideration once the tooling has been built and problems are experienced in production. However, implementing venting features on the tool structure after the tool has been built can be complicated and sometimes even impossible due to the space restrictions imposed by features and components such as cooling channels.

Venting issues do not only produce bad quality parts but can also put the mould tool itself at risk. The most familiar effect of poor venting on the part geometry is the burn mark (Figure 1), which actually shows up as a black spot on the plastic part. This black area is a thin carbon deposit created where the plastic is literally burned due to high temperature compressed gases trapped at the end of fill.

While the burn marks are most evident on the plastic component, the chemical action of these trapped gases at high temperatures can actually etch away the steel in these areas. This will eventually create an undercut, which will need to be corrected to avoid the part sticking during demoulding. The decision not to install a low cost venting pin or feature in a hard-to-fill area can end up costing several thousand Euros to rebuild a cavity.

Venting may be required not only to eliminate gas traps but also where weld lines occur or at very thin wall sections. Ensuring during the DFM process that venting features can be placed saves a lot of trouble as well as cost. Analysis tools such as flow study applications will help to determine sensible areas on the component structure where venting would be required.

Venting issues are most often created by the component geometry and Figure 2 shows an example where a rib design can be improved to ease the filling and avoid a gas trap on the flow path end simply by implementing a chamfer on the edge of the rib.

Having a good understanding of the tooling technology and venting requirements of moulding materials will help greatly during the design phase of the plastic component. Venting can be achieved in several ways, such as venting pins, ejectors, special designed core splits, or venting channels on the main parting lines which can be connected to the component geometry with small venting grooves. Three key factors need to be taken into consideration while setting up a proper venting system on a tool: dimensioning the vents, keeping them clear, and venting the gas.

Dimensioning the venting grooves

All moulding materials will have different flow behaviour so there will also be different requirements for the

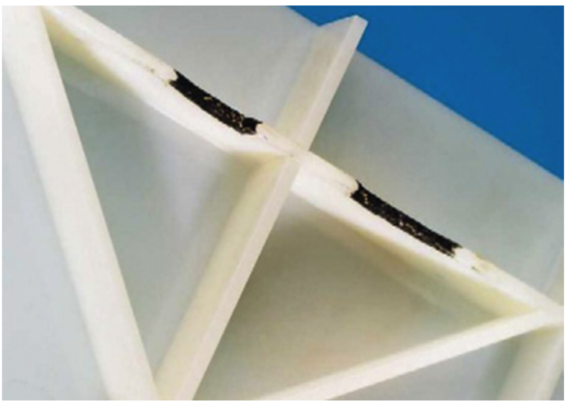


Figure 1: Burn marks caused on the ribs of this moulding are due to trapped gas

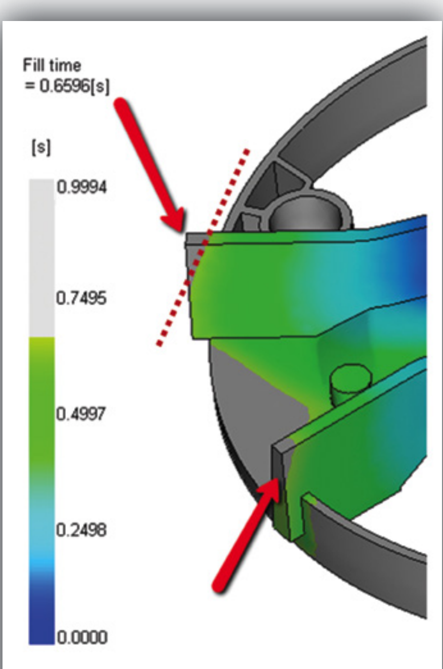


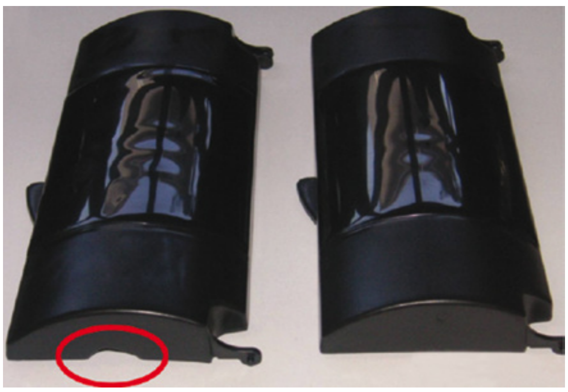
Figure 2: Flow analysis shows that adding a chamfer on the rib eliminates the gas trap with no need to vent

dimensions of the vents. We can create a small gap on the shut off area of the tool that is large enough to allow the gas to escape but not large enough to allow the part to flash. ABS/PC blends will generally allow good venting but no flash using a gap of 0.0012mm, whereas PA would require a smaller gap of 0.0008mm to avoid flashing problems. This is usually the reason why venting is considered at the design stage but not executed on the tool completely - the tool maker starts to open the venting grooves only after the first trials when venting issues will have been seen.

Keeping the vents clear

Cleaning of the venting grooves is essential to keep consistent part quality during production as the venting grooves and channels tend to get contaminated and close down after a period of operation. A maintenance schedule for cleaning appropriate to the material being used must be determined to prevent this (where glass fibre reinforced materials are involved the interval of cleaning stops will need to be shorter). Whatever the cleaning frequency, it is very beneficial if venting features can be accessed for cleaning without disassembling the whole tool or even while it is sitting on the moulding machine. The tool designer should always design a tool so that it can be cleaned easily and in a short time to minimize production interruptions. Ejectors can be considered to be self-cleaning as they move after each shot. Even complex steel features can move a little while the mould tool is opening or can follow the ejection stroke completely. During a recent mould optimisation project running a 50% glass reinforced PA it was found that by putting all the venting pins onto the ejector package it was possible to leverage the self-cleaning aspect to extend cleaning interruptions to 100,000 shots rather than the previous 30,000.

Figure 3: Incomplete filling of this part (left) was overcome using mould cavity evacuation technology from Frimo



Allow the gases to vent

This very important point is regularly missed. In some moulds all venting requirements may be taken care of by careful implementation of staggered venting inserts, pin-venting, venting grooves on insert splits and the like, but no consideration given to exhausting the gases, which will just compress somewhere within the tool structure. If the gases cannot escape freely to the outside atmosphere of the tool the venting performance will be impacted dramatically.

Some moulders have achieved good results on problematic parts using vacuum pumps to support cavity venting. Connecting a vacuum pump to exhaust drillings allows air to be removed from the cavity prior to injection of the polymer. The only additional element required is to install seals around the component geometry on the main parting line and on the ejector box to prevent air being sucked back into the cavity.

Figure 3 shows a component which could not be filled due to incorrect calculation of the maximum flow path during development. However, by evacuating the mould tool prior to injection it was possible to achieve a complete fill. The additional cost of the seals and vacuum pump components saved having to construct a complete new hot half with three drops to replace the original single drop version. It should be noted, however, that vacuum techniques should not be considered as a first option – the priority should always be placed on proper DFM work in the first place.

Cooling considerations

Cooling is generally seen as "the processing step" where most money can be saved and, with it accounting for 90-95% of a typical moulding cycle, it is the most attractive area to target to reduce cycle time. Shorter cycle times can make a big impact on production cost for a number of reasons:

- Fewer mould tools or cavities need to be manufactured to meet the anticipated production volumes;
- Smaller moulds can be used, allowing smaller and less costly moulding machines to be employed;
- Fewer moulds and cavities means labour cost can be reduced;

For the same reasons, the amount of energy required to product each part can be reduced.

A great deal can be done on the tool side to improve cooling time by implementing optimal cooling channels in the tool structure and taking care to achieve good heat transfer from the cavity into the steel and out of the tool via the cooling circuits. The biggest mistake in this area is to jump from large cooling channel diameter to a smaller diameter along the cooling channel ans this will create flow restriction.

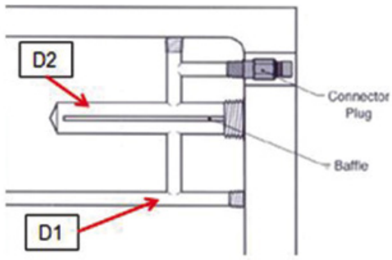
A simple mistake that is seen very often is to make bubblers or risers too small compared to the main cooling lines. Cutting a 10mm diameter riser hole in half will leave only 5mm diameter per side as flow channels. This mistake will leave areas of the tool with insufficient cooling and this will show up in poor component quality and dimensions, as well as extended cycle times. Figure 4 shows appropriate relationships between cooling channel and bubbler diameters.

To add cooling to the structure the most traditional and most often used technique is to drill into the tool steel. Unfortunately using this technique it is not possible to reach all areas of the tool as placement needs to be balanced with other tool features such as ejectors, gating, sliders, screw holes etc.

Manufacturing an insert in slices and milling the cooling channels in is quite an old and established technique. The individual steel slices are soldered together to form a solid insert and in this way it is possible to place cooling lines quite close to the component geometry. The downside is that additional witness lines may occur on the moulded component.

Better results in terms of short cycle time and high component quality can be achieved using laser sintered inserts. Laser sintering is a highly flexible manufacturing method that creates a 3D structure by fusing together layers of metal powder using a laser beam. It allows small cooling channels to be established very close to the cavity wall of the component geometry and the savings on cycle time can be phenomenal. AST has been involved in one project where a cycle time reduction up to 75% was achieved while also improving the dimensional and visual quality of the product.

However, there is also a risk attached to part cooling. Sometimes, high cooling rates or extended cooling periods are used to overcome warpage problems by



Ø Cooling Line D1	Ø Bubbler D2
Ø 6 mm	Ø 8 mm
Ø 8 mm	Ø 11 mm
Ø 10 mm	Ø 14.5 mm
Ø 12 mm	Ø 18 mm
Ø 14 mm	Ø 23 mm

Figure 4: Bubblers are often incorrectly sized to achieve proper cooling. This table shows some suitable diameter recommendations

freezing tension into a part. While this may reduce immediate post moulded distortion, it is an attempt to process around a component geometry issue such as a wall thickness difference along the flow path, incorrect gate point location or other root causes. And tension frozen in to the component by longer holding and cooling time may be released later on in the manufacturing process, such as where heat treatments are used.

Figure 5 shows a mobile phone cover that was treated in this way. A longer cooling time did lead to a flatter part but after the plating process, which applies a lot of heat to the component, the internal stress was released and showed up on the visible surface. Several component design changes were needed to modify the filling behaviour, as well as additional assembly and plating tests. The cost to modify the four production tools was calculated at close to €80,000 but the total financial damage caused by new testing slots, additional travel and labour costs, and delayed ramp up would have been much higher.

However, as cooling needs are determined by component geometry, this means that the component designer can influence this area very effectively. Analysis tools within CAD programs such as thickness analysis, cooling time plots and flow simulations will show the weak areas on a component design where cooling improvement is needed. Removing thick areas is the most important target to get the overall cooling time down, as well as improving the part quality.

The final instalment in this three-part discussion of mould design will be published in the May edition of *Injection World*.

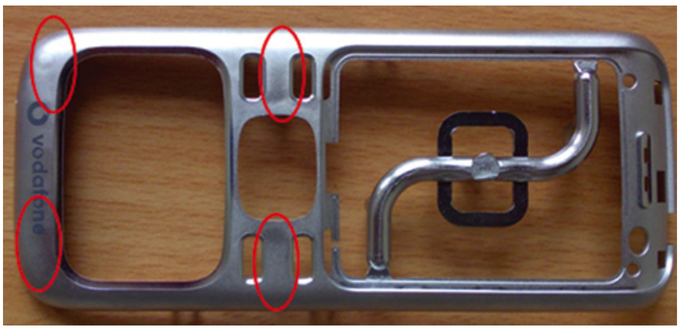


Figure 5: Release of frozen-in stress can cause problems in post moulding operations such as plating

About the author:

André Eichhorn is general manager of Germany-based AST Technology. This is the latest in a series of articles presenting a step-by-step discussion of the Design for Manufacturing (DFM) process. Previous instalments in this series can be viewed **here**, **here**, and **here**.

www.ast-tech.de