John Goff looks at the factors involved in filling an impression in the latest article in his series on how to achieve optimum injection cycles

Investigating melt flow

As the requirement to produce components to micron tolerances increases, the use of speed control becomes imperative. This article describes the basic elements that are important when filling an impression.

The flow behaviour of a molten thermoplastic material, when injected into a mould cavity via the gap between the surface of the core and the cavity wall, is often described as "fountain flow." This occurs when the molten material is forced along the gap and the outer layers come into contact with both metal surfaces. It then "freezes off" and stops flowing, insulating the flowing molten inner core as it is forced further along the gap to fill the impression. The resultant flow front is in the form of a parabolic curve, in which the maximum flow velocity is in the centre of the melt stream and the minimum velocity is at the interface with the metal surfaces (Figure 1).

This solidified skin (frozen layer) is progressively created as the impression is being filled. The material does not physically flow along the surfaces of the mould, but is forced against them. The thickness of the frozen layer tapers from the flow front back to a position along the flow path where it becomes constant. This position varies depending on the different shapes and complexity of the component as frictional energies are generated at the interface between the frozen layer and the molten core. The extent of frictional energy is determined by the velocity at which the molten material passes along the gap, that is, the injection speed used to produce the component.

The actual thickness of the frozen layer that is formed has a determining effect on the visual, structural and dimensional characteristics of the resultant moulding. In addition to injection speed, the thickness

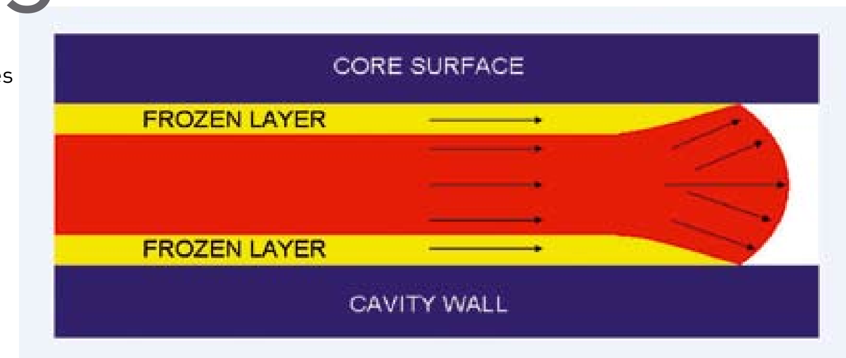


Figure 1: Parabolic curve

Top photo: SPIRAL FLOW INSERT, FOR AIM TEST MOULD, AXXICON, WWW.AXXICON.COM

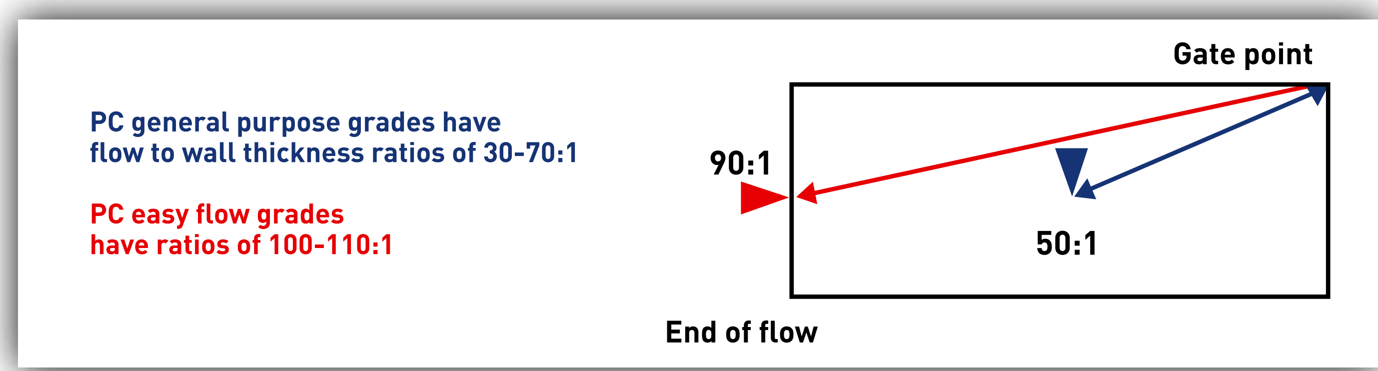


Figure 2: Example flow length:wall ratio for two grades of PC with a wall thickness of 1 mm

of the frozen layer is influenced by the selection of certain process parameters and the actual wall section of the moulded component. For this reason, the velocity of the melt flow front needs to be optimised in accordance with the thickness of gap (wall section); the length that the molten material has to flow from the gate point(s); and the type of thermoplastic material being processed and its related melt viscosity.

That is why thin walled components require the use of fast injection speeds to prevent premature freeze off taking place, to thereby prevent short and/or non-uniform surface finish mouldings. Furthermore, the longer the distance the molten material has to flow from the gate point(s) to achieve a full component, the greater its influence on achieving consistent component manufacture. The more effective the manner in which the mould cavity is filled, the more capable the process. Too many instances arise when the selection of process conditions causes the moulding process to be on a "knife edge" and result in the production of noncompliant, substandard mouldings that require regular minor modifications of process parameters to achieve conformity of component quality.

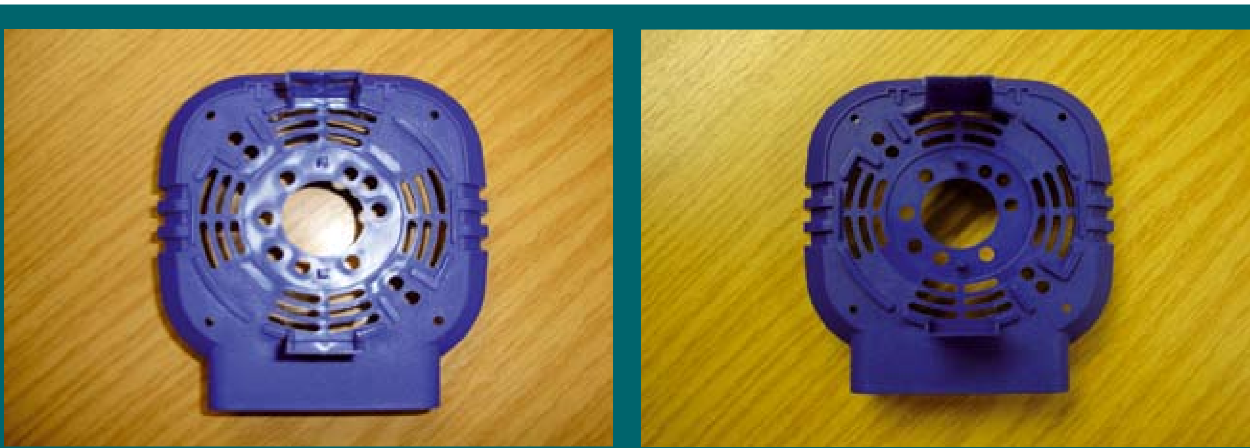
One important factor regularly overlooked is the selection of gate position. Each thermoplastic material has its own flowability rating with regard to the length at which it can effectively flow for a particular wall section. For each material, a flow length:wall section ratio value is given for a particular wall thickness.

Different material grades have varying flow length capabilities. The example in Figure 2 would only be filled using an easy flow grade of PC. What is also important is that simply increasing the wall thickness from 1 mm to 3 mm does not automatically increase the flow length threefold. More importantly, the thickness of the frozen layer can be greater than the typical 15% of the wall section per side, depending on the process parameters that are selected. For this reason, the design of the plastic component, together with the wall section used and the selected gate position are effectively optimised using computerised flow simulation software packages such as Moldflow, Moldex or Sigmasoft, to name a few. Table I gives some typical flow length:wall section ratio values for a 1-mm wall section.

Therefore, an understanding of the material's flow behaviour provides invaluable information for selecting the correct process parameters to produce the moulded component. Furthermore, the extent of friction derived at the interface of the frozen layer and the molten core on filling considerably affects the internal structure and consequential performance of the moulding when in service.

Good moulding practice advocates that the fullness of cavity achieved during the filling phase of the moulding process should be 95% to 98% by weight. This fullness provides a suitable baseline on which the applied holding

FIGURE 3: FULL MOULDING CONTAINING VISUAL SINKS (LEFT) AND A FULL DIMENSIONALLY AND VISUALLY COMPLIANT MOULDING (RIGHT)



(packing) pressure can be effectively introduced. For multicavity mould tools the variation in weight encountered across all impressions needs to be restricted to 5% or less. The lower the variation, the greater the consistency achieved as a result of the selected holding pressure being equally distributed within each impression to cater for the inherent shrinkage.

There is often a misunderstanding regarding 95% to 98% fullness. In reality, the moulding visually appears to be 100%, but it will contain sinks and/or ripples when viewed closely because of excessive shrinkage taking place during cooling. These sinks are a result of holding pressure not being applied. It is only when the holding pressure is applied in order to achieve a visually and dimensionally compliant moulding that its weight will increase. Figure 3 shows a moulding with pronounced sinks and a visually compliant moulding with perfect holes.

Quite often processors believe that the moulding should be partially short, thus representing 95% to 98% fullness. The components in Figure 4 show the difference between a 95% full moulding compared with one that is 92% full by weight.

Plastics materials by nature consist of numerous individual flexible chains, each containing chemical elements: carbon, hydrogen, oxygen and nitrogen or compounds containing groups of these. When these chains are stretched (in the state of tension), the resultant component has good integral physical properties, compared with mouldings produced with a structure possessing a high degree of compression.

In general, the faster the mould cavity is filled, the more consistent and capable the moulded components become. There is, of course, a limit to the extent to which the thermoplastic material is sheared when using fast injection speeds; too high an injection speed can be as detrimental as a too slow an injection speed. The next article in this series will discuss how to select the right melt flow velocity.

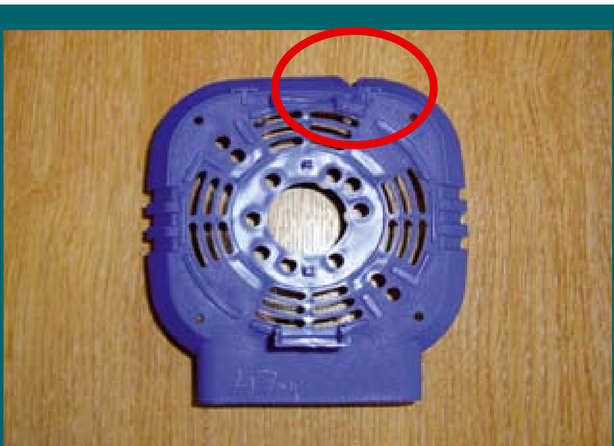
Table I: Typical flow length:wall section ratios for 1-mm wall section, range depends on the grade of material

Material	Abbreviation	Flow:Wall thickness
Acrylonitrile butadiene styrene	ABS	100 – 200:1
Acrylate styrene acrylonitrile	ASA	180 – 230:1
Ethylene vinyl acetate	EVA	200 – 300:1
High impact polystyrene	HIPS	250 – 340:1
High density polyethylene	HDPE	200 – 270:1
Low density polyethylene	LDPE	200 – 300:1
Linear low density polyethylene	LLDPE	180 – 250:1
Nylon (polyamide) 6	PA6	160 – 300:1
Nylon (polyamide) 66	PA66	180 – 300:1
Nylon (polyamide) 11 & 12	PA11/PA12	180 – 220 1
Polybutylene terephthalate	PBT	140 – 220:1
Polycarbonate	PC	30 – 110:1
Polyetherimide	PE1	70 – 140:1
Polyethylene terephthalate	PETP	220 – 350:1
Polyethylene terephthalate glycol	PETG	50 – 90:1
Polymethylmethacrylate	PMMA	110 – 170:1
Acetal copolymer	POM-CO	100 – 250:1
Acetal homopolymer	POM-HO	100 – 250:1
Polypropylene	PP	230 – 340:1
Polyphenylene oxide	PPO-M	100 – 200:1
Polyphenylene Sulphide	PPS	120 – 185:1
Polystyrene (general purpose)	PS/GPPS	150 – 200:1
Styrene acrylonitrile	SAN	170 – 200:1
Sulphone polymers	PSU	60 – 120:1
Polyether sulphone	PES	60 – 140 :1
Plasticised polyvinylchloride	PPVC	200 – 250:1
Unplasticised polyvinylchloride	UPVC	80 – 190:1

More information

This is the tenth article in the Moulding Masterclass series, which discusses the fundamental issues that prevent optimal injection cycles. Recent articles can be accessed, here, here and here, respectively. John Goff is Managing Director of G&A Moulding Technology.

www.gandamoulding.co.uk



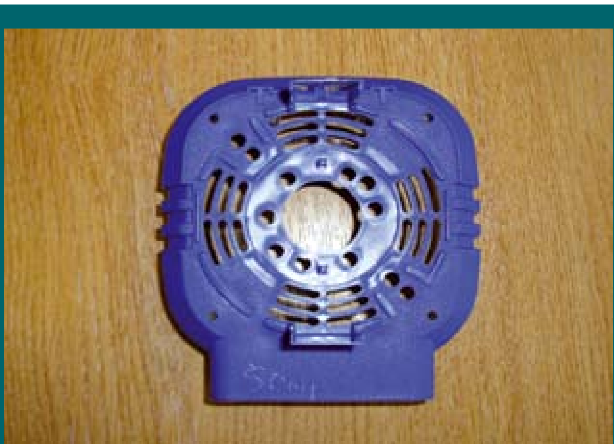


FIGURE 4: 92% FULLNESS (LEFT) WITH PIECE MISSING; 95% FULLNESS (RIGHT), THE SHAPE OF THE MOULDING COMPLETE