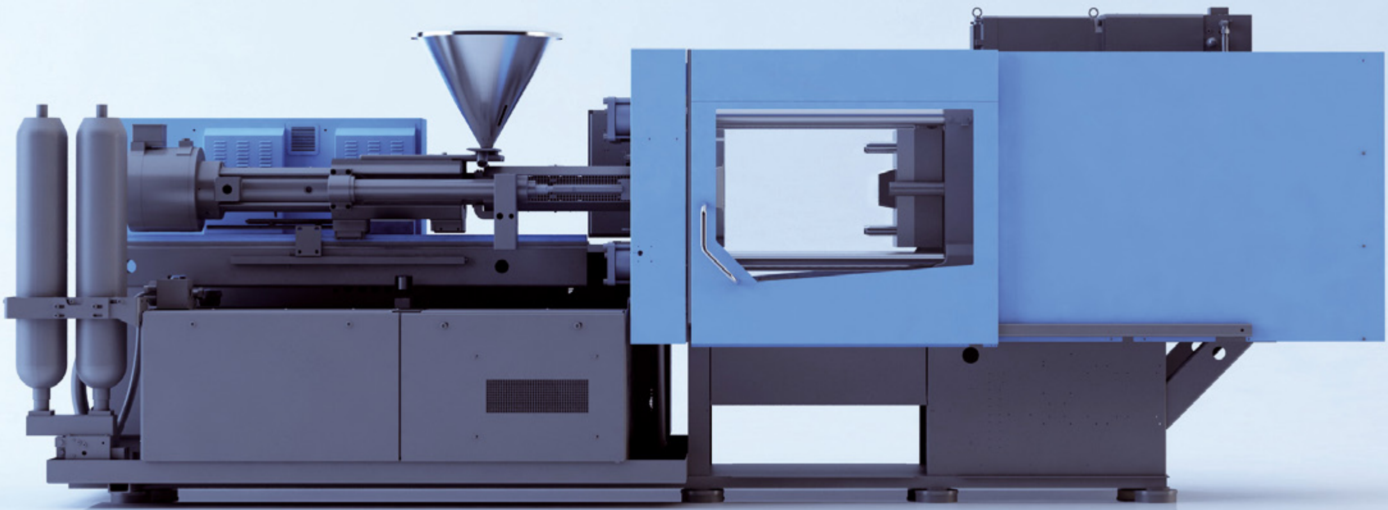


John Goff discusses why it is important have a fast injection time in the latest article in his series on injection moulding process optimisation

Optimum injection time

As briefly covered in the **last article** in this series, when selecting an injection speed to produce a moulded component, the actual speed used should be as fast as possible to suit the material being processed and the shape and complexity of the component. The word "fast" becomes an important expression, and over the years there have been numerous discussions, sometimes rather heated ones, on what is the best injection speed to employ.

Plastics materials by nature contain many flexible chains that need to be manipulated/orientated to achieve optimum flowability. The selected injection speed must ensure that the material's structural characteristics are not affected during processing. As a general rule, injection moulded components that possess a molecular structure in which the orientated chains are in tension (stretched) exhibit better mechanical, optical and end performance properties than those in which the structure is mainly in compression.

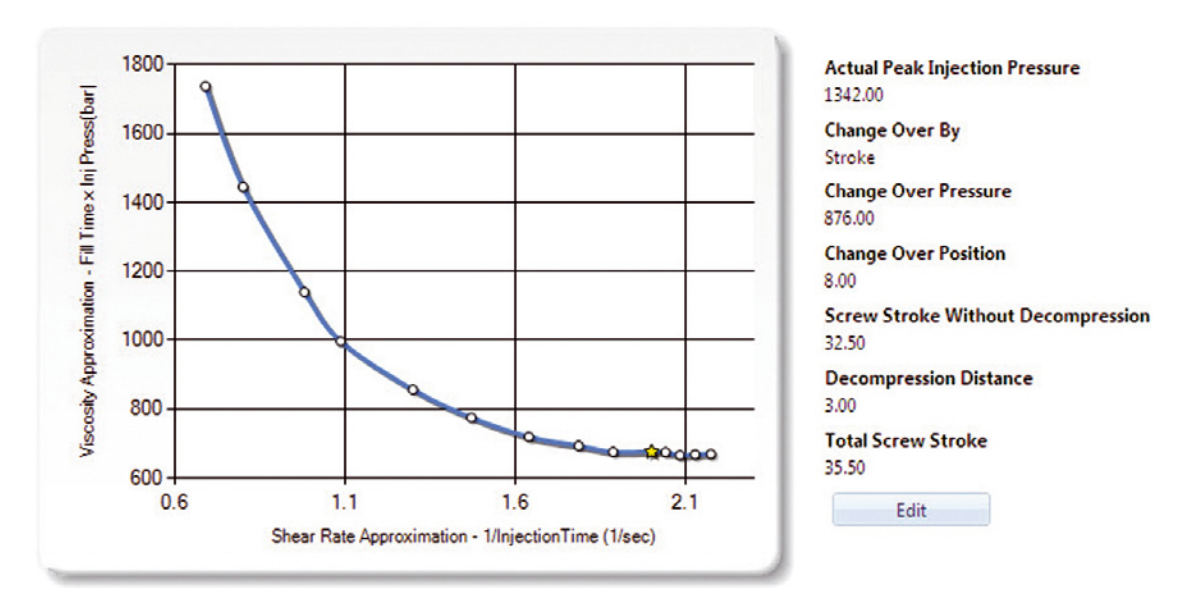
The injection speed that is selected to produce the component dictates the molecular structure that is created. Hence, the correct selection of this value is very important for both process consistency and moulding capability.

Test for correct speed

A simple birefringence test will demonstrate the importance of using the correct injection speed. When using a fast injection speed the molecular chains are stretched in the line of flow, whereas with a slower injection speed the molecular chains are compressed and result in a different structural condition. When viewing the clear moulded component between polarised filters the light rays that pass through the filters and moulding are suitably detected. Any change in their wavelength and passage because of internal refraction and/or reflection, in accordance with the level of internal stress contained within the moulding, is detected by the colours of the spectrum from red, blue, green, yellow, indigo and violet present in the moulding.

Furthermore, the closer the bands and intensity of colour, the greater the levels of light interference and

VISCOSITY SHEAR RATE CURVE



consequently levels of stress. When the molecular chains are in tension, the absence of the spectrum colours denotes a less stressed or relatively stress free component.

Usually the dominant colour found is black when the chains are in tension. When the molecular chains are formed in a compressive condition, the extent of the colours denotes the level of internal stress present.

Naturally, too fast an injection speed is as detrimental as one that is too slow, where the internal structure (molecular chains) of the moulding becomes damaged due to excessive shear heat (this point will be discussed in a separate article in this series). When selecting the injection speed value for a particular mould tool–plastics material–machine combination, various demands are placed on the processor to achieve the required aesthetic, dimensional and functional quality standards.

The most popular mode of control used globally for the filling stage of the cycle is by volume, stroke or position. Hence, the injection speed selected for a given screw stroke to achieve 95% to 98% fullness can be directly related to the time it takes for the screw to move forwards from the screw stop (dosing) position to the changeover/switchover position. Put simply, the faster the injection speed, the shorter the time value and vice versa.

For correctness of moulding performance and process parameter monitoring, it is the injection time not injection speed that is important. Transfer of process parameter settings from one moulding machine to another would evolve around time values. This latter point is extremely important when a validated moulding process has to be transferred from one machine to another, even if the machine is of the same make and type.

Before considering the available techniques for achieving correct the injection time value for a particular moulded component, thought must be given to air removal. Uppermost in the processor's mind should be how the contained air within each cavity/core set and feed system for a cold runner mould tool is removed as the molten polymer fills the feed system and subsequently the impression.

The importance of air removal

It is true to say that a significant percentage of all mould filling issues evolves around the manner in which the injection time is selected to accommodate effective removal of air. Manipulation of the injection time value is often performed in an attempt to overcome faults such as burn marks, shortness of components, distorted components and surface finish issues due to inadequate venting being available on the mould tool. These time changes often lead to moulded components possessing flow marks, poor surface integrity, poor optical quality and highly stressed parts, to name a few defects.

Inadequate removal of air from the impression is a common problem and, contrary to common belief, air removal is equally troublesome with small mouldings or large components, in particular with two-shot or two-component moulding. The presence of air can and often does cause hesitation (slowing down) in the melt flow front velocity as the material fills the impression and leads to other quality issues such as flow defects, poor weld line(s) and/or poor weld line strength.

When determining the optimum injection time for a

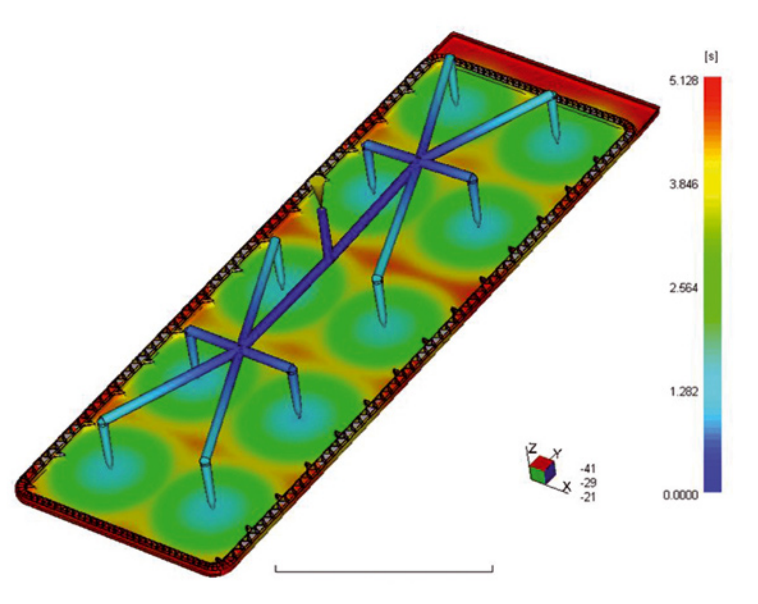


PHOTO: PROMOLDING

CAE SOFTWARE PACKAGES CAN PLAY AN IMPORTANT ROLE IN OVERCOMING GAS ENTRAPMENT ISSUES

particular moulding, therefore, good mould tooling practices need to be employed. It is common practice within injection moulding industry for a vent to be purposely left out by the mould maker before the mould tool has been sampled in the belief that only during sampling can the actual position be determined to effectively allow the air to escape.

Although this practice has good principles it does have disadvantages. The processor sampling the mould tool often encounters the presence of air/gas entrapment and will then slow the injection speed down to overcome the burning/gassing issue and therefore not truly address the issue. Furthermore, on qualification of the mould tool and start of the production run, the problem of gassing returns, which leads to process parameter changes taking place and inconsistent product quality. Time wasting discussions can occur with claims that the problem was not present during the sampling, and questions asked as to why it should be present now!

Often overlooked is the fact that although elimination of burning and/or shorting occurs with injection speed reduction, the air that had been previously compressed due to the melt flow front viscosity is still present within the mould cavity. This air is forced backwards into the moulding itself. Often mechanical failure occurs to the component at the end of flow where the air accumulates and becomes entrapped. On viewing this under a microscope, the wall section at point of failure is often found to be porous, with a foam like structure, which indicates that the air has not been fully removed during the filling phase.

Furthermore, when the processor reduces the injection speed process, instability will occur. This is often encountered when a material batch or colour change takes place due to viscosity variation. It occurs within the polymer melt as it flows into the cavity and leads to problems of dimensional variability and/or quality issues. When determining the optimal injection time, therefore, it is imperative that certain mould tool attributes are interrogated, in particular, venting, gate size, gate location and feed system design (both cold and hot runner mould tools). It is essential that any issues involved with these are addressed before the mould tool is put into production.

Effective prediction of gas entrapment can now be undertaken using CAE software packages such as Moldflow, Moldex 3D and Sigmasoft, to name a few. However, it must be said that within the compendium of packages offered not all of them predict where the gas entrapment occurs when using mould filling simulation.

Determining injection time tends to be based on previous knowledge and experience. With the introduction of the computerised injection moulding machine, which accurately collects and displays the actual process data, the optimal injection time can be technically derived to complement experience. In general, the injection times derived solely through experience tend to be longer than those obtained through data acquisition and technical derivation.

The value of data

Because the viscosity (flowability) of a thermoplastic material is predominantly related to the temperature selected for processing and the applied shear (injection speed) generated to fill the mould tool cavity, a simple relative viscosity/shear rate curve can be created for each mould tool–material–machine combination. Creation of this curve by conducting a series of process changes provides the processor with invaluable information regarding the capability of the injection moulding machine and correctness of performance from the mould tool. The flow curve created, as shown in the figure, pictorially displays the manner in which the molten material is behaving during the filling of the mould cavity.

Techniques to use to achieve correct injection time will be examined in the next issue of *Injection World*.

More information

This is the eleventh 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