Numerical simulation of the filling phase in the polymer injection moulding process with a conservative level set method

R. El Otmani[1], M. Zinet[1], M. Boutaous[1], P. Chantrenne[1], H. Benhadid[2]

[1]*CETHIL, UMR 5008, CNRS, INSA Lyon, Université Lyon1 – F-69621 Villeurbanne, France*
e-mail: rabie.el-otmani@insa-lyon.fr; mhamed.boutaous@insa-lyon.fr

[2]*Laboratoire de Mécanique des Fluides et d'Acoustique, UMR 5509, CNRS, Ecole Centrale de Lyon, Université Lyon1, INSA Lyon - F-69134 Ecully Cedex, France*
e-mail: hamda.benhadid@univ-lyon1.fr;

ABSTRACT: Numerical simulations are widely used in polymer processing in order to understand the interaction between the material properties and the process. In the injection moulding process, one of the numerical simulation problems encountered is the tracking of the polymer-air front or interface during the filling stage [1]. In this study, this is achieved by using the classical level set method, which reveals to be conservative. On the contrary, in the case of confined volume of two phases flow, such as an air bubble moving in a constant volume of liquid, the method needs a correction in order to be conservative [2-3]. For a non-Newtonian non-isothermal flow, the simulation involves a coupling between heat transfer, continuity and momentum equations and the rheological material parameters. A finite element method is used to solve this equations set. The results show clearly the expected fountain flow effect. Temperature, velocity, pressure and viscosity fields are calculated and the influence of the thermal contact resistance between the polymer and the mould is presented.

Key words: Injection moulding, Level set method, Polymer, Heat transfer.

1 INTRODUCTION

A numerical model is developed to simulate the filling stage of the injection moulding process. In order to obtain an accurate prediction of the filling flow behaviour we have combined a conservative level set method with a momentum viscous flow equations.
The first section of the paper concerns the level set method applied to simulate a bubble rising in a liquid. In this case, the classical level set method is not conservative, and a correction is introduced to insure the mass conservation of the system.
In the second section this method is applied to describe the front evolution during filling in the injection moulding process. The mathematical model of the flow and the level set method is briefly introduced. In this case, the proposed correction is not necessary, because, contrary to the bubble flow configuration, the air volume in the injection moulding is escaped and replaced by the polymer liquid. The simulation of a polymer flow in a rectangular cavity is then performed in non-isothermal conditions.
In the last part, a parametric study of the filling flow is presented and discussed. Fountain flow, temperature profiles, density and velocity evolutions are presented and commented. The role of the thermal contact resistance (TCR) is shown by comparing simulation results with and without a representative TCR value.

2 LEVEL SET METHOD

2.1 The level set equation

In order to track the interface between two fluids in two phases flows, several methods exist in the literature. The standard level set method is known to be easy to implement numerically. But in several situations, this method needs a correction in order to be conservative. Here, we present an application of this method to follow the boundaries of an air bubble moving in a liquid (water). It consists in a gas bubble rising in a cylindrical volume of water, where the interface energy or tension plays an important role. It is then necessary to compute correctly this

interface. The volume of the whole system remains constant, but the resolution of the problem by means of the level set method shows that the bubble loses a part of its volume when moving. A correction based on a penalty factor is then proposed to insure the volume conservation.

The flow problem is modelled by the following equation:

$$\rho\frac{Du_i}{Dt}=\frac{\partial\left(\eta\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right)}{\partial x_j}-\frac{\partial p}{\partial x_i}+\rho g_i \quad (1)$$

$$+\sigma k_i\frac{\partial\phi_i}{\partial x_j}\delta(\phi_i)$$

where D/Dt denotes the material derivative, σ is the surface tension, $\delta(\phi)$ is the Dirac function, and k_i the curvature defined by:

$$k_i=-\frac{\partial N_i}{\partial x_j} \quad (2)$$

with N_i the normal vector defined as:

$$N_i=\frac{\partial\phi_i/\partial x_j}{\left|\partial\phi_i/\partial x_j\right|} \quad (3)$$

The interface location is determined implicitly by means of the level function ϕ by solving the following equation:

$$\frac{\partial\phi_i}{\partial t}+u_i.\frac{\partial\phi_i}{\partial x_j}=u_i.\frac{\partial^2\phi_i}{\partial x_j{}^2} \quad (4)$$

where $\phi = 0$ correspond to the interface.

In equation (4) the right side term corresponds to a numerical added viscous term.

In order to compute the distance function ϕ, an initialisation equation is introduced and iterated for few steps in fictitious time τ:

$$\frac{\partial\psi_i}{\partial\tau}+S(\phi_0)\left(\left|\frac{\partial\psi_i}{\partial x_j}\right|-1\right)=0 \quad (5)$$

where $S(\phi_0)$ = sign(ϕ_0) and $\psi(t = 0) = \phi_0$. see [4] for more details concerning the level set function.

To compute the material properties (density, heat capacity, thermal conductivity, viscosity) at the junction between the two fluids, one introduces a Heaviside function:

$$H(\phi)=\begin{cases}0, & \phi<0\rightarrow \quad fluid1\\ 1, & \phi>0\rightarrow \quad fluid2\end{cases} \quad (6)$$

In numerical simulations, the abrupt jump of the fluids properties due to equation (6) causes instabilities when using the finite element method. Therefore, a smeared out Heaviside function is introduced:

$$H_{sm}(\phi)=\begin{cases}0, & \phi<-\varepsilon,\\ \frac{1}{2}+\frac{\phi}{2\varepsilon}+\frac{1}{2\pi}\sin\left(\frac{\pi(\phi)}{\varepsilon}\right), & -\varepsilon\le\phi\le+\varepsilon\\ 1, & \phi>+\varepsilon\end{cases} \quad (7)$$

where ε corresponds to the half thickness of the interface or an arbitrary chosen small number. The interface thickness shall depend on the mesh elements size. The thermophysical properties are then linked to ϕ.

In a system with two immiscible incompressible fluids, they can be defined by using classical mixture laws:

$$g=g_1+(g_2-g_1)H_{sm}(\phi) \quad (8)$$

where g_1 is defined in Ω_1 (polymer domain), and g_2 in Ω_2 (air domain).

2.2 *Correction with the penalty factor*

Due to the numerical approximations, a mass loss occurs in the flow. Corrections are then needed to insure a mass conservation. To do so, we introduce an absolute constraint in the kinematic condition:

$$\frac{1}{2}\beta\left(\int_\Omega H_{sm}(\phi)\,d\Omega-Vol^*\right)^2 \quad (9)$$

where β is the penalty factor, and Vol^* is the initial volume of the disperse phase.

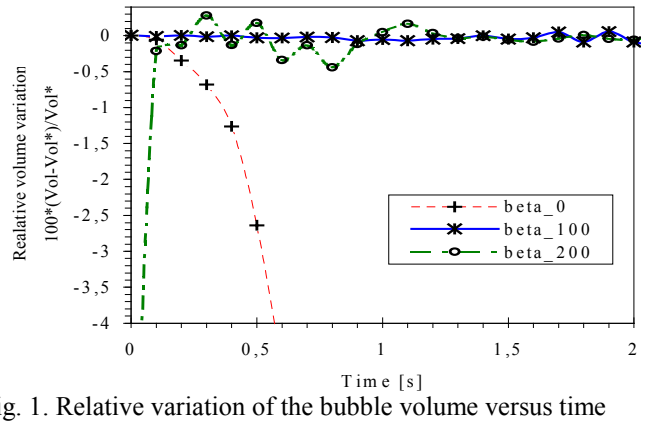

Fig. 1. Relative variation of the bubble volume versus time

Figure 1 shows the relative variation, in percent, of the air bubble volume versus time. Two cases are confronted. The first one without using penalty factor ($\beta = 0$), the second one by using penalty factor (β = 100, 200). We observe that the introduction of a penalty factor enhances clearly the volume conservation. Particularly, in this case, the β = 100 is the optimal value.

3 FILLING FLOW AND ENERGY MODEL

3.1 Non-isothermal viscous fluid flow equations

Momentum, continuity and energy equations for incompressible flows can be respectively written as.

$$\rho \frac{Du_i}{Dt} = \frac{\partial\left(\eta\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right)}{\partial x_j} - \frac{\partial p}{\partial x_i} \tag{10}$$

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{11}$$

$$\rho c_p \frac{DT}{Dt} = \eta.(d_{ij} : d_{ij}) + \frac{\partial}{\partial x_j}\left(k\frac{\partial T}{\partial x_j}\right) \tag{12}$$

where D/Dt denotes the material derivative, ρ, c_p, and k are the density, the heat capacity and the thermal conductivity of the fluid respectively. The strain tensor d is defined as:

$$d_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{13}$$

To effectively describe the shear thinning effect, a Cross viscosity law is employed:

$$\eta\left(T,\dot{\gamma},p\right) = \frac{\eta_0(T,p)}{1+\left(\frac{\eta_0(T,p)\dot{\gamma}}{\tau^*}\right)^{1-n}} \tag{14}$$

where the equivalent strain rate is given by:

$$\dot{\gamma} = \sqrt{2d_{ij} : d_{ij}} \tag{15}$$

Here, n is the power-law index, τ^* the relaxation time, and $\eta_0(T, p)$ the zero shear rate viscosity.

3.2 Boundary and initial conditions

A no slip condition is imposed at the interface between the polymer and mould. Downstream the polymer flow front, a free slip condition is prescribed, in order to enable the contact point (air-polymer-mould) to move freely. Hence, the boundary condition along the mould walls is a function of the type of material, which is indicated by the level set function ϕ. The mould walls are impermeable, except at the air vents Γ_{out}, where the air is allowed to leave the mould. As an initial condition for the temperature problem, a uniform temperature field over the entire domain is imposed. At the injection gate Γ_{inlet}, an injection temperature is prescribed. At the walls, we assume a thermal contact resistance (TCR) between the polymer melt and the mould about 10^{-5} and 10^{-3} m^2.K/W.

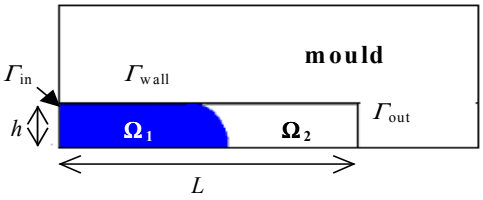

Fig. 2. Geometry of the mould cavity

The level set method presented in section 2 is coupled with the momentum equation without any correction. The standard formulation is then solved together with the flow and energy equations.

In figure 3, we have compared the polymer mass evolution in the cavity, from the initial time to the end of the filling step. Step by step during calculation, no differences are observed between the mass calculated and that ideally injected for each time. The level set method is then considered conservative.

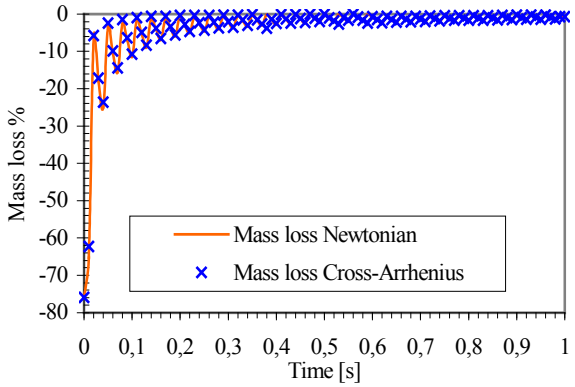

Fig. 3. Mass loss percentage versus filling time at the inlet of the cavity

4 RESULTS AND DISCUSSION

The polymer used is a polyacetal POM-M90-44. All its characteristics are given in [5]. The viscosity is represented by means of a Cross-Arrhenius Law.

Figure 4 shows the viscosity evolution at several locations in the mould with and without TCR. As expected, we can observe that the viscosity values are affected both by the temperature and the shear rate. The introduction of a TCR influences the viscosity evolution as a consequence of its effect on the heat transfer. Near the mould walls, the viscosities are higher due to the temperature effect modeled by the Arrhenius law.

In Figure 5, we present the cross-sectional temperature distributions at several longitudinal positions X. One notices an important temperature gradient from the wall to the core. This gradient is less important when a TCR is taken into account,

and a discontinuity in the temperature profile is then observed. This temperature profile is responsible for all the thermophysical properties gradients in injected parts. The high viscosity values near walls, due to the decrease in temperature, leads to the reduction of the velocities in this zone. Hence, because of the mass conservation, the flow is accelerated in the central zone (far from the walls) as shown in figure 6. The no-slip boundary condition at the mould walls and the polymer-air interface cause a stretching flow and a shear flow in this zone. The front continuously splits along the interface, leading to a 2D velocity field in the thickness direction. This phenomenon is called fountain flow and is well represented by the model. Figure 7 shows this effect on the polymer front at several times during the filling stage. The fountain flow is responsible, in large part, of the physical properties of the skin of injected parts.

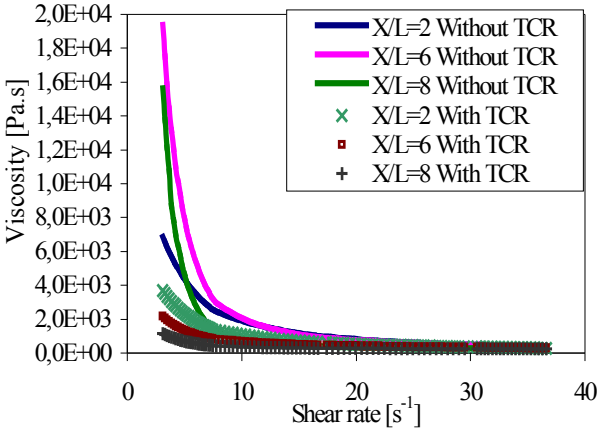

Fig. 4. Viscosity versus shear rate at the end of the filling time for several locations *X/L* in the cavity: influence of the TCR

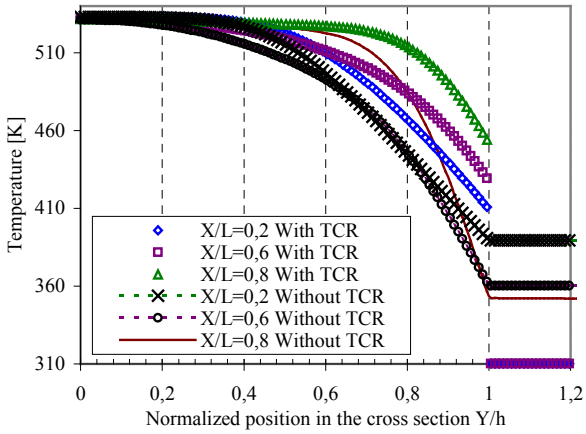

Fig. 5. Temperature field within the cross section *Y/h*, at the end of the filling time, for different locations *X/L*: influence of TCR parameter

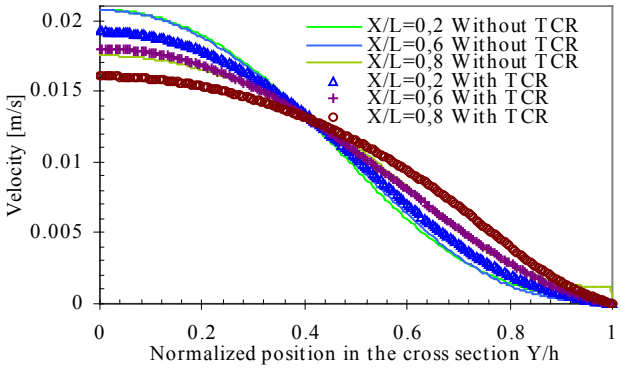

Fig. 6. Velocity profile within the cross section *Y/h* for different location *X/L*, influence of TCR parameter

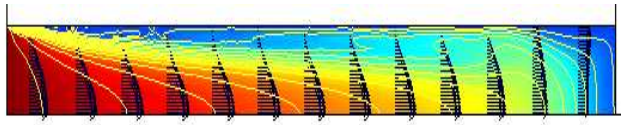

Fig. 7. Fountain flow (isocontours) and velocity field (arrows)

5 CONCLUSIONS

In this paper numerical experiments demonstrate and validate the performance of the level set method in describing interface evolution in two phases flows. In confined volumes, a correction is needed but still easy to implement. An example is performed in the case of a rising bubble. This method is thus suitable for the numerical simulation of filling phase in the polymer injection moulding process. A parametric study of this configuration confirms that all the physical phenomena can be represented with good accuracy.

REFERENCES

1. G. A. A. V. Haagh, H. Zuidema, F. N. Van de Vosse, G. W. M. Peters, and H. E. H. Meijer, "*Towards a 3-D finite element model for the gas-assisted injection moulding process*", Int. Polym. Proc., 1997, Vol.12, p.207-215.
2. E. Olsson and G. Kreiss, "A conservative level set method for two phase flow", J. Comput. Phys., 2005,Vol. 210 (1), p.225–246.
3. Sethian J. A., "Numerical algorithms for propagating interfaces: Hamilton-Jacobi equations and conservation laws", J. Diff. Geom., (1990), Vol.31, p. 131-161.
4. J. A. Sethian, Numerical methods for fronts, in Variational methods for free surface interfaces, Eds. P. Concus and R. Finn. Springer-Verlag, New York (1987). (2005).
5. R. El Otmani, M. Zinet, M. Boutaous, P. Chantrenne, H. Benhadid, Numerical Simulation of the filling stage in the polymer injection moulding process, Proceedings of the European COMSOL Conference, volume 1 315-320 (2007).