DROPLETS IN MICROCHANNELS WITH LEVEL SET METHOD

Paul Vigneaux
Bordeaux 1 University, Mathematiques Appliquees de Bordeaux
351, Crs de la Liberation, 33405 Talence Cedex, France
e-mail: Paul.Vigneaux@math.u-bordeaux1.fr

Key words: Level Set, Microfluidics, droplets

Abstract. The field of microfluidics has dramatically improved over the last decade: it enables the transport of minuscule amounts of fluid (nanolitres) through channels the diameter of a single human hair. This miniaturization is increasingly employed to build a wide range of technological devices for example, genome sequencing or using microdroplets as chemical reactors.

The aim of this paper is to introduce a two-dimensional numerical model and approximations of bifluid flows in microchannels in order to simulate the dynamics of microdroplets. The bifluid model is derived following the level set approach of [5]. We perform a numerical stability analysis which leads to a restrictive condition on the computational time step. We then present a method to relax this constraint. We give details and numerical results of the drop forming due to pinching and the evolution of droplets in microchannels using the presented method.
1 INTRODUCTION

The field of microfluidics has dramatically improved over the last decade: it enables the transport of minuscule amounts of fluid (nanolitres) through channels the diameter of a single human hair. This miniaturization is increasingly employed to build a wide range of technological devices for example, genome sequencing [1] or using microdroplets as chemical reactors [2]. But a better understanding of the evolution of droplets as they move through microchannels is necessary in order to achieve the flow control needed in practical applications. The Applied Mathematics Laboratory of Bordeaux (MAB) has developed a numerical model that simulates the dynamics of a droplet in microchannels. This model can be used to complement physical studies on droplet dynamics. The advantage of a numerical model is that it allows rapid simulation of experiments so that various interface configurations can be considered before verifying with longer, more costly physical experiments.

The paper is organized as follows. In next section, we describe the physical problem and assumptions. We then present the mathematical formulation considered. The model is derived following the level set approach of Sussman et al. [5]. In section 3, we detail a numerical stability analysis for our model involving surface tension forces. This leads to a restrictive condition on the computational time step. Then, the section 4 presents a method which allows to relax the stability condition on the time step. Thus, this level set method shortens simulation times. Finally, section 5 is devoted to the application of this method to drop forming. Numerical results are performed to show the ability of the method to handle droplet dynamics in microchannels.

2 PHYSICAL AND MATHEMATICAL FRAMEWORK

2.1 Physical considerations

We are working in collaboration with physicists who conduct experiments of multifuid flows in microchannels. In particular, they use cross-channel geometries in order to make droplets of one fluid into another. In microfluidics, such process can then be repeated to obtain reproducible drop size and shape due to an excellent control of the flow. This is a way to produce monodisperse, multiple emulsions [2].

We want to simulate unsteady immiscible bifuid flows in such microchannels. This allows us in turn to compare the results with the physical studies of our colleagues. For the applications we are considering, the Reynolds number is lower than one. We can also assume that fluids are incompressible, viscous and Newtonian. Thus, we can use a quasi-stationary Stokes equation to model the flow. The viscosities and densities are constant in each fluid since they are supposed to be homogeneous. We adopt a sharp-interface approach in which the interfacial thickness is zero as opposed to diffuse-interface approach (e.g. phase-field methods). As a consequence, viscosities and densities exhibit
a jump discontinuity at the interface. We further assume that the interface has a surface tension with a constant coefficient.

It is interesting to note that even if the Reynolds number is low, the flow is not strictly laminar. This is due to the fact that, at these scales, surface tension forces are preponderant over inertia forces. As a consequence, even if the structures in the flow are simple (e.g. droplets) there may be strong vortex inside these structures. As we will see in sections 3 and 4, this also induce a special care to advance computation in time via a stability condition on the time step.

As exposed above, we also have to deal with moving interface and topological changes since we want to be able to create droplets in our microchannels. Bearing in mind that, in this paper, we want to stay in a two dimensional framework to make our simulations, we will need to provide a special treatment of the surface tension term as we will discuss later in section 5.

It has been shown over the past 15 years, that Level Set methods [3] [4] are particularly designed for problems in which topology of the interface changes during the simulated evolution. Due to its Eulerian nature, Level Set method allows to easily handle topological changes of the interface. We thus adopt a Level Set method to simulate droplet forming and evolution in our microchannels.

2.2 Mathematical formulation

We base our mathematical formulation upon the level set method proposed by Sussman et al. [5] for incompressible two-phase flows. The Level Set method was initially devised by Osher and Sethian in [6] for computing the motion of an interface in any dimension.

The idea is to represent a given interface $\Gamma \subset \mathbb{R}^d$ as the zero isocontour of a continuous function $\phi : \Omega \subset \mathbb{R}^d \to \mathbb{R}$. Suppose now, that the interface is moving under the velocity field $u$. The Level Set method consists in obtaining the evolution of the interface by solving the classic convection equation:

$$\phi_t + u \cdot \nabla \phi = 0 \quad \forall x \in \Omega, \forall t > 0$$

(1)

The Level Set function $\phi$ is defined on the whole domain $\Omega$. This is a front capturing method since the interface evolution is embedded in the evolution of the higher dimensional implicit function $\phi$. At any time $t$,

$$\Gamma(t) = \{ x \in \Omega / \phi(x,t) = 0 \}$$

(2)

In practical applications, it has been shown that taking $\phi$ as a signed distance function (such that $|\nabla \phi| = 1$) leads to better accuracy in the results [7]. The velocity $u$ results in the embedding of the interface’s velocity to a higher dimension velocity field also defined
on $\Omega$. It is a work on its own to give a meaning of $u$ throughout $\Omega$ and not just on the interface itself, where it is obvious. This is achieve by specific techniques of construction of extension velocities [3] [4].

In addition to topological changes which are handled naturally, another advantage of level set formulation is that if the velocity depends on intrinsic geometric quantities of $\Gamma$ such as the normal vector $n$ or the curvature $\kappa$, these quantities are easily obtained with the level set function, since :

$$ n = \frac{\nabla \phi}{|\nabla \phi|} \text{ and } \kappa = \text{div}(n) $$ (3)

We follow the formulation of [5] and adapt it it to our microchannels flows. We use the Stokes equation with surface tension to compute the velocity and pressure fields. This velocity is then used to move the Level Set function. In other words :

$$ \text{div}(2\eta Du) - \nabla p = \sigma \kappa \delta(\phi)n $$ (4)  
$$ \text{div}(u) = 0 $$ (5)  
$$ \phi_t + u.\nabla \phi = 0 $$ (6)

where $Du = (\nabla u + \nabla^T u)/2$, $\eta$ is the variable viscosity such that :

$$ \eta = \begin{cases} 
\eta_1 & \text{in fluid 1} \\
\eta_2 & \text{in fluid 2}
\end{cases} $$ (7)

$\sigma$ is the surface tension coefficient, $n$ is the normal to the interface, $\kappa$ is the curvature of the interface and $\delta(\phi)$ is the Dirac function which allows to localise the interface since the level set function $\phi$ is defined as the signed distance function to the interface :

$$ \phi \begin{cases} 
> 0 & \text{in fluid 1} \\
= 0 & \text{on the interface} \\
< 0 & \text{in fluid 2}
\end{cases} $$ (8)

2.3 Numerical considerations

We realize all our numerical simulations in the 2D case. The domain geometry is composed of channels which cross each other with injections of various fluids at various extremities, or exits of fluids, depending on the physical experiments. (see Figure 1)

In addition, we must impose slipping boundary conditions (BC) on the walls of the channels :

$$ \begin{cases} 
u \tau = \alpha u_s(\eta) + \beta L_s(\eta) \frac{\partial(u,\tau)}{\partial n_w} \\
u.n_w = 0
\end{cases} $$ (9)
where \( n_w \) and \( \tau \) are respectively the normal and the tangent to the boundary, \( u_s \) is the slip velocity and \( L_s \) is the so called “slip length” coefficient both depending \textit{a priori} on \( \eta \) since slipping is different from one fluid to another. We suggest the hybrid condition (9) to take into account whether the interface touches the wall or not. Classical boundary conditions are used at the inlets and outlets.

![Figure 1: An example of geometry](image)

In order to solve the Stokes equation (4), we use a finite volume discretization on staggered cartesian grid and an Augmented Lagrangian method to take into account the incompressibility constraint (5). The transport equation (6) is solved through the fifth order WENO schemes [8].

3 NUMERICAL STABILITY ANALYSIS

We now recall the numerical algorithm we use to advance the interface forward in time, in order to introduce the stability constraint, induced by discretization, on the time step.

At the beginning of the simulation, the fields \( \eta \) and \( \phi \) are given, \textit{i.e.} we know the initial interface. Then we iterate the following steps, up to the desired time \( t_{end} \):

- using \( \phi^n \), compute its normal \( n \) and curvature \( \kappa \) needed for the Stokes problem ;
- compute the velocity field \( u^n \) solution of the Stokes problem (4)-(5) ;
- compute the new level set function \( \phi^{n+1} \) thanks to the transport equation \( \phi_t + u^n.\nabla \phi = 0, u^n \) and \( \phi^n \).

From a formal point of view, there is a highly non-linear coupling between the velocity field \( u \) and the level set function \( \phi \) since \textit{via} the Stokes equation, \( u \) depends on the second derivatives of \( \phi \) - say formally, \( u = \text{Operator}(\Delta \phi) \) - and \textit{via} the transport equation : \( \phi_t + \text{Operator}(\Delta \phi).\nabla \phi = 0 \). We then clearly understand that the discretization time step
must satisfy a non-trivial stability condition.

This is a well known issue for a long time now. Nevertheless, we just give an illustration of the consequence of the use of a standard CFL condition as stability condition for the above algorithm. In Figure 2, we show successive interfaces obtained when the initial interface is an unconfined ellipse in a microchannel flow. We observe, this droplet rapidly attains a circular shape - thanks to the surface tension forces - but then remains stationary in the channel (in the meaning, even if the shape of the interface is modified, the centroid of the droplet has a nearly fixed location in the channel) and oscillates around this perturbed circular shape. Note that the correct computation, should provide an almost circular interface moving along the channel due to the injection forcing and not this still perturbed circle. Actually, the observation of the velocity fields associated to these oscillating interfaces show huge velocities (compared to the injection forcing) near the interface (see also Figure 3). These effects can be related to what is classically known in the literature as parasitic currents.

![Figure 2: Stability problem : green ellipse is the initial interface, in black are the following interfaces forward in time ; the green vertical lines are the wall of the channel](image)

As noted before, the need for a specific stability constraint on the time step has been introduced for a long time. Following again Sussman et al., a time step must be considered for each contribution of the model. The first we have to think of, is obviously the CFL condition induced by the convective terms. Then, other terms of the flow equation induced other restrictions on the time step. In their approach, they have to consider stiff source terms like gravity, surface tension and viscous terms. In particular, they follow the stability condition of Brackbill et al. [9] for the surface tension term. But all these derivations were for the Navier-Stokes equation.

The problem of parasitic currents and stability condition on the time step appear to be linked as we shall describe this now.
In the following, we present a derivation of a stability condition for our Stokes Problem. The time step associated to the convection is classically \( \Delta t_c = \Delta x / \max(u) \). We therefore concentrate on the surface tension term of (4).

### 3.1 Capillary-wave stability analysis

The problem illustrated by Figure 2 is due to the fact that only the classical CFL condition is considered. It allows the propagation of the interface over a distance the order of the space step \( \Delta x \). This leads to the stationary behaviour of the interface since it oscillates around the asymptotic shape induced by the surface tension, i.e. a nearly circle shape (or a perfect circle if there is no injection). Actually, since the interface is not the asymptotic one, the velocities found by the Stokes problem are huge - they are induced by the surface tension source term which acts to move the interface towards the circular shape (more precisely, the shape which minimize the surface energy at the interface) - and the CFL condition induces \( \Delta t_c = \Delta x / \max(u) \), thus the interface move of a distance of \( \Delta x \) and the time step is smaller than the one associated to the injection (see Figure 3). By the way:

- there is no stabilization of the interface since, at each iteration, it always moves of \( \Delta x \), from one side to another of the asymptotic location;
- the interface is globally “stationary” since it oscillates around asymptotic shape with tiny time steps.

To fix this problem, we have to take into account the velocity induced by surface tension when the interface exhibits a perturbation of the asymptotic shape. The idea behind the stability condition is to control the motion of the interface, in such a way it does not cross the asymptotic location. We detail the steps of the derivation of a stability condition.

In fact, the problem illustrated by Figure 2 also takes place when there is no injection. Thus, we first consider an interface in a flow without injection.
First step : reformulation of the source term

The Stokes equation can be rewritten as:

\[
\text{div}(2\eta Du) - \nabla p = \sigma \kappa_0 \delta(\phi) n
\]
\[
= \sigma \kappa \nabla H(\phi)
\]
\[
= \sigma \nabla(\kappa H(\phi)) - \sigma H(\phi) \nabla \kappa
\]  \hspace{1cm} (10)

since \( \delta(\phi) n = \nabla H(\phi) \) and we made an extension of the curvature \( \kappa_0 \) of the interface \( \Gamma \) over the whole considered domain \( \Omega \):

\[
\kappa = \begin{cases} 
\kappa_0(x) & \text{if } x \in \Gamma \\
\kappa(x) & \text{if } x \in \Omega \setminus \Gamma 
\end{cases}
\]  \hspace{1cm} (11)

In a Level Set framework, this extension can be done by solving the equation \( \kappa_t + n \cdot \nabla \kappa = 0 \). Gathering the gradient terms, we get:

\[
\text{div}(2\eta Du) - \nabla (p + \sigma \kappa H(\phi)) = -\sigma H(\phi) \nabla \kappa
\]  \hspace{1cm} (12)

Remark 1 This formulation (10) has several advantages. First, in the case of a perfect circular interface, we have \( \nabla \kappa = 0 \) and then the source term is a pure gradient. By the way, it follows directly that the unique solution of the Stokes problem is \((u, p + \sigma \kappa H(\phi)) = (0,0) \) and thus

\[
p = -\sigma \kappa H(\phi) = \begin{cases} 
-\sigma \kappa & \text{if } \phi > 0 \\
0 & \text{if } \phi < 0 
\end{cases}
\]  \hspace{1cm} (13)

The surface tension is responsible for the pressure jump at the interface and the velocity is equal to zero in the whole domain. Second, in the perspective of a perturbation analysis, we will see in the following that this formulation allows us to show that a small perturbation of the interface leads to a small perturbation of the velocity.

We now want to estimate the source term \(-\sigma H(\phi) \nabla \kappa\) in order to get an estimation of the velocity.

Second step : curvature of a perturbed interface

We have to consider a special case of a perturbed interface so that we can evaluate the term \( \nabla \kappa \). For ease of presentation, let us consider the following perturbed interface:

\[
f(x) = \delta \cos \left( \frac{2\pi}{L} x \right)
\]  \hspace{1cm} (14)

The curvature of \( f \) is given by:

\[
k(x) = \frac{f''(x)}{(1 + (f'(x))^2)^{3/2}}
\]  \hspace{1cm} (15)
We want to estimate the gradient of the curvature. In this purpose, we look for the maximum of \( k' = \frac{dk}{dx} \) which depends on two parameters: \( \delta \), the perturbation amplitude and \( L \), the wavelength of the perturbation. Note that in the perspective of discretization, one should think of \( L \) as the minimum resolvable wavelength i.e. \( 2\Delta x \) and of \( \delta \) as the maximum perturbation which can induce instability i.e. \( \Delta x \) (since if \( \delta > \Delta x \), then, thanks to a CFL condition, the interface does not move farther along a distance of \( \Delta x \) and thus it does not cross the asymptotic location). In addition \( f \) is a periodic symmetric function; we thus just need to search the maximum on \( (x, \delta) \in [0, L/2] \times [0, L/2] \). Some calculus leads to:

\[
\max_{x \in [0, L/2]} k'(x) = \frac{C(\delta)}{L^3} \delta \quad \text{for} \quad \delta \in [0, L/2]
\] (16)

with \( C(\delta) \) such that:

\[
\lim_{\delta \to 0} C(\delta) = 8\pi^3
\] (17)

\[
\lim_{\delta \to L/2} C(\delta) = D
\] (18)

where:

\[
D = 64 \frac{\sqrt{6 \pi^2 + 5 - \sqrt{3} \sqrt{12 \pi^4 + 16 \pi^2 + 7} \pi^2 (-3 + \sqrt{3} \sqrt{12 \pi^4 + 16 \pi^2 + 7})}}{L^3 (9 + 6 \pi^2 - \sqrt{3} \sqrt{12 \pi^4 + 16 \pi^2 + 7})^{3/2}} \approx 680
\] (19)

This computation show that \( C(\delta) \) does not degenerate neither for \( \delta \to 0 \) nor \( \delta \to L/2 \).

**Third step : estimation of the induced velocity** An analysis of (12) leads to the following estimation:

\[
\frac{\eta u_{\text{pert}}}{\Delta x^2} \sim \sigma |\nabla \kappa|
\] (20)

where \( u_{\text{pert}} \) is the velocity induced by the perturbation of the interface defined by \( f \). From (16), the gradient of the curvature is controlled by:

\[
|\nabla \kappa| \leq \frac{C(\delta)}{\Delta x^3} \delta
\] (21)
we then have the estimation:

\[ u_{\text{pert}} \sim C(\delta) \frac{\sigma \delta}{\eta \Delta x} \]  

\[(22)\]

**Remark 2** Tanks to the formulation (10), and taking into account that \( C(\delta) \) is bounded, it follows:

\[ \lim_{\delta \to 0} u_{\text{pert}} = 0 \]  

\[(23)\]

**Fourth step : a stability condition**  The stability condition induced by the surface tension term is design in such a way the perturbation does not cross the asymptotic shape (see Figure 5) i.e.

![Diagram showing perturbed and asymptotic interfaces with \( \delta \) denoting the perturbation thickness.](image)

**Figure 5:** Stability condition linked to surface tension

\[ \Delta t_{\sigma} \leq \frac{\delta}{u_{\text{pert}}} \]  

\[(24)\]

Plugging (22), we finally get:

\[ \Delta t_{\sigma} \leq \frac{\eta}{\sigma} \frac{\Delta x}{C(\delta)} \]  

\[(25)\]

**Remark 3** We can do the same analysis with the unsteady Stokes equation:

\[ \rho u_t - \text{div}(2\eta Du) + \nabla (p + \sigma \kappa H(\phi)) = \sigma H(\phi) \nabla \kappa \]  

\[(26)\]

Taking into account that the unsteady term and the source term are of same amplitude and plugging (21), it successively follows:

\[ \rho \frac{u_{\text{pert}}}{\Delta t} \sim \sigma |\nabla \kappa| \quad \text{i.e.} \quad u_{\text{pert}} \sim \frac{\sigma}{\rho} \frac{\Delta t}{\Delta x^3} C(\delta) \delta \]  

\[(27)\]

Hence (24) becomes:

\[ \Delta t \leq \sqrt{\frac{\rho}{\sigma C(\delta) \Delta x^3}} \]  

\[(28)\]
where $\langle \rho \rangle$ is the mean of $\rho$ since the density is a variable density defined on the whole considered domain, as the viscosity field. This stability condition is of the type of the one derived in [9].

3.2 Summary

The condition (25) was derived for a flow without injection. It can be extended to the case where there is injection forcing by considering the splitting of the velocity $u = u_{in} + u_{pert}$ where $u_{in}$ is the velocity associated to the injection forcing and $u_{pert}$ is the one associated to the surface tension.

$$\phi_t + u_{in} \cdot \nabla \phi + u_{pert} \cdot \nabla \phi = 0$$

For the former, the stability condition is $\Delta t_c = \Delta x / \max(u_{in})$ et for the later, the stability condition is (24). Finally the computational time step is given by :

$$\Delta t = \min(\Delta t_c, \Delta t_\sigma)$$

We must also note that the condition associated to surface tension is somehow relaxed when it is used on the discretized problem. This can be understood, in the Level Set framework, by computing the highest curvature in the discrete form : it is of order $1/\Delta x^2$. Actually, in practical computation, it is more likely :

$$\Delta t_\sigma \leq \frac{\eta}{\sigma} \Delta x$$

A complete justification of this stability analysis is done in [10].

3.3 Microfluidics applications

Let us consider our microfluidics applications. We have $\eta = 10^{-3} - 10^{-2}$ Pa.s and $\sigma = 10^{-3} - 10^{-2}$ N/m. At the inlet, the injection velocity $u_{in} = 10^{-2}$ m/s. By the way :

$$\Delta t = \frac{\Delta x}{u_{in}} = 100 \Delta x$$

$$\Delta t = \frac{\eta}{\sigma} \Delta x = \Delta x$$

As a consequence, we see that for these applications, the standard CFL is by far greater than the stability condition induced by surface tension. This explains the instability seen in Figure 2. In addition, our numerical tests show that (31) gets rid of the instability.

However, we also note that (31) is very restrictive compared to the CFL since it is hundred times smaller. In the following, we propose an approach to relax this stability constraint.
4 A LEVEL SET APPROACH FOR MICROFLUIDICS

Let us consider a stabilized interface in a convective flow. During numerical simulation, we want to advance forward in time in a less restrictive way than the one imposed by (31). In this purpose, we introduce a splitting which take into account the two motions associated to the injection forcing and the surface tension forces.

First, let us expose the following consideration: in microfluidics, the droplets are often moving with a stationary shape (e.g. when the channel is straight). The idea consists in avoiding work which is unnecessary to do ; when a droplet is only in a translation motion in the channel, we do not want to advance forward in time with a such restrictive stability condition as (31). To this end, we work in the droplet’s frame of reference using a splitting.

The procedure is as follows:

1. Find the speed of translation of the droplet and the time step $\Delta t_{inj}$ associated to the injection

2. Compute the shape correction of the interface with an iterative step for which the total time is less or equal to $\Delta t_{inj}$ with a combination of $\Delta t_{\sigma}$ time steps. The stationary shape is then obtained

3. Compute the translation of the interface by solving the transport equation over the time step $\Delta t_{inj}$ and the resulting speed induced by speeds computed in (2)

4. Loop (2)-(3)

This splitting allows to shorten simulation time because in (b) the stationary shape is obtained with less iterations than $N = \frac{\Delta t_{inj}}{\Delta t_{\sigma}}$. Also, one can advance faster forward in time by translating this stationary shape up to the moment for which the interface becomes unsteady (e.g. when, after a straight channel, the interface enters a bend.

The details of the explanation of the efficiency of this splitting are given in [10].

4.1 Numerical results

We consider the case of an initial ellipsoidal droplet in a convective flow in a microchannel. The droplet is small enough to stay unconfined even when it reaches its asymptotic shape. The velocity is taken in such a way the surface tension forces are prominent. Due to surface tension, the asymptotic shape of the droplet is almost a circle.

The channel has a width of 200 $\mu$m. The injection velocity $u_{inj} = 8.10^{-3}$m/s. The viscosity of the droplet is $\eta_1 = 10^{-3}$Pa.s and the viscosity of the continuous phase is $\eta_2 = 10^{-2}$Pa.s. The surface tension coefficient $\sigma = 5.10^{-2}$N/m.

The Figure 6 shows the evolution of this droplet using the splitting method. We see that there is no instability, contrary to Figure 2.
Figure 6: Simulation of an unconfined ellipse. Top left: initial (blue) and successive (black) interfaces superimposed. Top right to Bottom right: successive interfaces at $t = 1.7$, 3.4 and 17 ms which show the convergence of the shape towards the nearly circular asymptotic. The same part of the channel is shown for all figures.

5 APPLICATIONS TO DROP FORMING

5.1 The problem

As presented previously the geometry used is composed of channels which cross each other. In practical applications, such kind of microfluidic device is used to create droplets, as it is shown on Figure 7.

However, one can not simulate drop forming with a pure 2D code as the one presented above. This is due to the fact that the jet developing in the exit channel is always stable in the 2D case (see e.g. [11]). Indeed, all perturbation of a 2D jet causes an increase of the surface area of the jet (i.e. an increase of the surface energy) which is always controlled by surface tension forces. Actually, it is the 3D extension of the jet, say, a cylinder of fluid, which is unstable because there exist perturbations which can cause a decrease in surface area (see e.g. [12]). In these cases, the surface tension induces the breaking of the
jet and creates droplets since it is a way to lower the surface energy at the interface of the two fluids.

Nevertheless, we want to be able to obtain droplets with our 2D model, in order to observe the behaviour of this level set approach when topological changes occur. This is achieved using the following trick. We transform the 2D curvature term (computed with $\phi$) in a pseudo-3D one by considering the axisymmetrical extension of our 2D jet. Namely, instead of considering the 2D curvature

$$\kappa = \text{div}(n)$$

we introduce the 3D curvature

$$\kappa = \text{div}(n) + \frac{1}{R_2}$$

where $R_2$ is the radius of curvature in the orthogonal plane of the computational plane (see Figure 8). This radius $R_2$ is obtained by considering the jet is axisymmetrical. By the way, it suffices to evaluate the width of the jet in order to get the value of $R_2$. 

Figure 8: Extension of the 2D curvature. Blue arrows are the surface tension forces in the computational plane. Green arrows are the surface tension forces in the orthogonal plane.
This is easily achieved with the level set function $\phi$. The curvature (35) shows that, we now have a correction term which can induce instability. Indeed, $1/R^2_2$ has always the same sign whereas the sign of $\text{div}(n)$ depends on the curvature of the interface in the 2D computational plane. Thus when the jet is narrowing, $1/R^2_2$ can change the sign of the global curvature term and then induces the break of the jet.

5.2 Numerical results

We consider a jet which develops from the upper vertical channel into the exit channel. The initial interface and the flow configuration are presented in Figure 9.

![Figure 9: Initial interface and flow configuration](image-url)

The channel has a width of 50 $\mu$m. The injection velocity $u_{inj} = 10^{-2}$m/s. The viscosity of the jet is $\eta_1 = 10^{-3}$Pa.s and the viscosity of the continuous phase is $\eta_2 = 10^{-2}$Pa.s. The surface tension coefficient $\sigma = 5.10^{-2}$N/m.

We compute two types of jet evolution. One with the 2D surface tension term (34), the other with the 3D surface tension term (35). Time evolutions of the jet are presented respectively on the left and right side of Figure 10. The view is the same in both cases and is centered on the location of the drop forming. This allows to compare the different behaviours induced by 2D and 3D curvature formulation. We observe numerically that the “2D jet” is stable contrary to the 3D one, in accordance with previous energy analysis.

The small asymmetry observed on the interface of the breaking jet is due to the computation of the curvature which can lose a bit of accuracy when the jet is narrowing.
Figure 10: Simulation of a microfluidic jet. Left: 2D surface tension, the jet does not break. Right: 3D surface tension, droplets are formed.
6 CONCLUSION

In this paper, we have presented a two-dimensional numerical model for bifluid flows in microchannels. A numerical stability analysis showed restrictive time steps arise in these applications and a splitting method was introduced to shorten simulation times. Finally a pseudo-3D model for surface tension was designed in order to simulate drop forming with only a 2D code. Actually, the developed splitting method, originally built for microfluidics, can be generalized for other applications and methods of interface capturing [10].

REFERENCES


