



HAL
open science

A sharp Cartesian method for incompressible flows with large density ratios

Michel Bergmann, Lisl Weynans

► **To cite this version:**

Michel Bergmann, Lisl Weynans. A sharp Cartesian method for incompressible flows with large density ratios. [Research Report] RR-8926, INRIA Bordeaux. 2016, pp.23. hal-01331234v1

HAL Id: hal-01331234

<https://inria.hal.science/hal-01331234v1>

Submitted on 13 Jun 2016 (v1), last revised 27 Mar 2017 (v2)

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



A sharp Cartesian method for incompressible flows with large density ratios

M. Bergmann¹, L. Weynans¹

¹Team Memphis, INRIA Bordeaux-Sud-Ouest & CNRS UMR 5251,
Université de Bordeaux, France

**RESEARCH
REPORT**

N° 8926

Juin 2016

Project-Teams Memphis



A sharp Cartesian method for incompressible flows with large density ratios

M. Bergmann¹, L. Weynans^{1*}

¹Team Memphis, INRIA Bordeaux-Sud-Ouest & CNRS UMR 5251,
Université de Bordeaux, France

Project-Teams Memphis

Research Report n° 8926 — Juin 2016 — 23 pages

Abstract: A new Cartesian method for bifluid incompressible flows with high density ratios is presented. The specificity of the method relies on a sharp second order numerical scheme for the spatial resolution of the discontinuous elliptic problem for the pressure. The Navier-Stokes equations are integrated in time thanks to a fractional step method based on the Chorin scheme and discretized in space on a Cartesian mesh. The bifluid interface is implicitly represented using a level set function. The numerical tests show the improvements due to this sharp method compared to classical first order methods.

Key-words: Incompressible flows, bifluid flows, finite-differences, projection method, cartesian grid, level-set, jump conditions across interface, interface unknowns

* Corresponding author: lisl.weynans@inria.fr

**RESEARCH CENTRE
BORDEAUX – SUD-OUEST**

351, Cours de la Libération
Bâtiment A 29
33405 Talence Cedex

Une méthode cartésienne précise sur l'interface pour des écoulements incompressibles avec de grands ratios de densité

Résumé : Nous présentons une nouvelle méthode cartésienne pour des écoulements incompressibles bifluïdes avec de grands ratios de densité. La spécificité de la méthode repose sur l'utilisation d'un schéma numérique non régularisé et d'ordre deux pour la résolution du problème elliptique discontinu pour la pression. Les équations de Navier-Stokes sont intégrées en temps grâce à une méthode à pas fractionnaire basée sur le schéma de Chorin, et sont discrétisées en espace sur une grille cartésienne. L'interface entre les fluides est représentée implicitement par une fonction level-set. Les tests numériques montrent les améliorations apportées par cette nouvelle méthode comparée aux méthodes d'ordre un classiques dans la littérature.

Mots-clés : Écoulements incompressibles, écoulements bifluïdes, différences finies, méthode de projection, grille cartésienne, level-set, conditions de saut au travers de l'interface, inconnues d'interface

Contents

1	Introduction	4
2	Governing equations	5
3	Numerical method outside the discontinuities	7
3.1	Interface computation	7
3.2	Flow computation	8
4	Specific sharp method for the discontinuities across the interface	9
4.1	Viscous terms	10
4.2	Divergence of predicted velocity	11
4.3	Elliptic problem near the interface	11
4.4	Pressure gradient near the interface and correction step	12
5	Numerical resolution of elliptic problems with immersed interfaces	12
5.1	Discrete elliptic operator	13
5.2	Discrete flux transmission conditions	13
6	Numerical validation	14
6.1	Parasitic oscillations	14
6.1.1	Comparison with the Ghost Fluid and the CSF methods	15
6.1.2	Comparison with a Volume of Fluid method	15
6.2	Rising of air bubble in water	17
6.2.1	Comparison with the Ghost-Fluid method	17
6.2.2	Comparison with SPH [10] and the level-set method [17]	20
6.3	Collapse of a water column (dam break)	20
7	Conclusion	20

1 Introduction

In this paper we are concerned with incompressible bifluid flows like air and water, and by the accurate description of the phenomena occurring at their interface. Our study aims to provide a new numerical method for the simulation of incompressible flows with high density ratios (and possibly with high viscosity ratios also) on Cartesian grids. Consequently, we present a sharp Cartesian method for the simulation of incompressible flows with high density and viscosity ratios. This method is inspired from the second-order Cartesian method for elliptic problems with immersed interfaces developed in [5].

Cartesian grids are an attractive alternative to body fitted meshes. Indeed, they avoid complex mesh generation as well as mesh adaptation when unsteady interfaces are considered. The numerical resolution of the governing equations can also be simplified thanks to an easy parallelization and the use of standard linear algebra libraries. Generally speaking, numerical schemes are easy to implement on a Cartesian mesh because a dimensional splitting is often possible. However, some numerical modeling is necessary near a complex interface that does not fit the background Cartesian grid. This is the case for fluid structure interface and moreover for bifluid interface where the properties of the flow are discontinuous. Indeed, applying naively a numerical schemes originally devised for a flow with constant or continuously varying density will lead to a non-consistent treatment of the interface. Most of the time, it will result in severe stability issues if the density ratio is large as highlighted in [21] and references therein. Therefore, as already mentioned, one has to devise specific numerical schemes at the vicinity of the interface. This region called narrow band is the set of numerical points that have at least one neighbor on the other side of the interface.

Conservative or non-conservative approaches can both be used to face this issue. Among the non-conservative approaches, one solution is to regularize in the vicinity of the interface the properties of the fluids, so that the density, viscosity, and their derivatives are continuous in the whole computational domain. This idea leads to the well known "Continuous Surface Force" method [2], where the discontinuous quantities are smoothed near the interface, and in case of a fluid with surface tension, this surface tension is taken into account as a smooth volume force instead of a surface force. This method is widely used (see for instance [17] and [9]) because it offers a straightforward way to implement the presence of two fluids in an already existing monofluid Navier-Stokes code. However, the exact way that the regularization should be performed is not always clear, and spurious oscillations at the bifluid interface can appear, due to errors in the pressure gradient computations. Another non-conservative method introduced by Kang, Fedkiw and Liu [13] after the CSF is the Ghost fluid method. It is based on a first order method developed in [14] to solve an immersed interface elliptic problem, with a dimensionnal splitting making the method easy to implement. The resulting linear system is symmetric and has the same structure as the usual matrix to discretize a Poisson equation with variable coefficient on a Cartesian grid. This method has been used successfully in numerous works, for instance [6] and [31]. One drawback is that the method is only first-order accurate near the interface [21] and a loss of conservativity of the momentum of each fluid near the interface can occur, and lead to erroneous velocities. Non-conservative methods are often associated with a level-set representation of the interface, as in [17], because the level-set method is itself intrinsically non-conservative at the discrete level, and convenient to use on a Cartesian grid.

The other family of methods is based on the conservative form of the Navier-Stokes equations, where mass and momentum fluxes of each fluid are explicitly computed, see for instance [23], [29], [30], [12] and more recently [21]. An explicit interface representation is necessary even if the interface do not coincide with grid points. Conservative methods are generally more stable than non-conservative methods, at the price of an increased amount of work due to this interface reconstruction, which can be performed from informations carried by Lagrangian markers or by cell quantities such as volume fractions.

In this paper we aim to preserve as much as possible the simplicity of the Ghost-Fluid method of [13], avoiding an explicit identification of the volume fractions near the interface, while improving the accuracy and stability of the pressure computation. We thus propose a new method where the discontinuities across

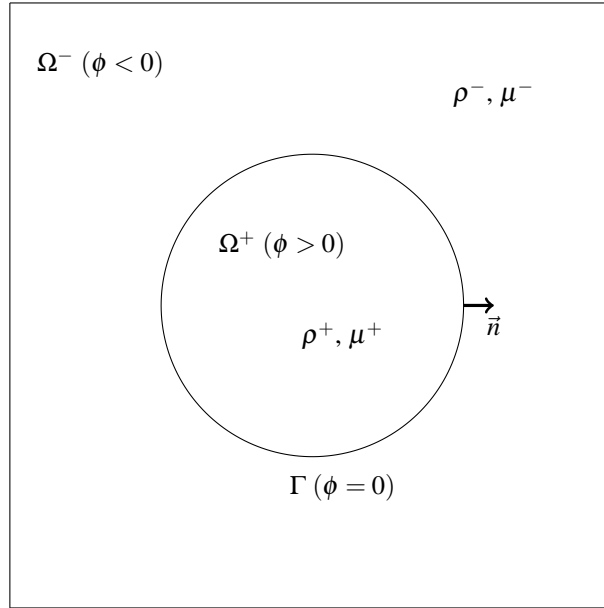


Figure 1: Sketch of the computational domain.

the interface are taken into account in a sharp way with a second-order scheme inspired from [5]. This second-order treatment improves the conservativity of the method, as it will be proved numerically in the section devoted to numerical validations.

After having described the governing equations for the incompressible bifluid flows that we consider (§2), the discretization of these equations in each fluid and at the interface are presented (§3-4). The second-order numerical resolution of the elliptic problem arising from the computation of the pressure is introduced (§5), and the overall is validated on several numerical test cases (§6).

2 Governing equations

Flow configuration We consider a rectangular domain Ω filled with two viscous incompressible fluids with distinct densities and viscosities. The subdomains Ω^- and Ω^+ corresponding to the two fluids are separated by an interface Γ . These domains are generally defined thanks to a scalar function ϕ that takes different values in each subdomains with a fixed value on the interface. For instance we chose $\phi = 0$ on Γ , $\phi > 0$ in Ω^+ and $\phi < 0$ in Ω^- . The unit normal to the interface is \mathbf{n} and the unit tangent vector is $\boldsymbol{\eta}$. The density is

$$\rho = \rho^- + H(\phi)(\rho^+ - \rho^-),$$

and the viscosity is

$$\mu = \mu^- + H(\phi)(\mu^+ - \mu^-),$$

where H is the Heaviside function, *i.e.* $H(\phi) = 1$ if $\phi > 0$ and $H(\phi) = 0$ if $\phi < 0$. Finally, the two-dimensional velocity vector is $\mathbf{u} = (u, v)$.

In fluid subdomains The flow is modeled in each subdomain with the incompressible Navier-Stokes equations:

$$\begin{aligned}\rho(\mathbf{u}_t + (\mathbf{u} \cdot \nabla)\mathbf{u}) &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g}, \\ \nabla \cdot \mathbf{u} &= 0,\end{aligned}$$

with \mathbf{g} the gravitational acceleration vector, and $\boldsymbol{\tau}$ the viscous stress tensor:

$$\boldsymbol{\tau} = \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T).$$

Jump conditions at the interface The above equations are completed by jump conditions at the interface Γ between the two fluids. In what follows jumps are defined by $[\psi] = \psi^+ - \psi^-$.

- The first ones describe the balance between the normal stresses at the interface and the surface tension σ , with κ the local curvature of the interface Γ ,

$$[p - 2\mu(\nabla \mathbf{u} \cdot \mathbf{n}, \nabla \mathbf{v} \cdot \mathbf{n}) \cdot \mathbf{n}] = \sigma \kappa, \quad (1)$$

$$[\mu(\nabla \mathbf{u} \cdot \mathbf{n}, \nabla \mathbf{v} \cdot \mathbf{n}) \cdot \boldsymbol{\eta} + (\nabla \mathbf{u} \cdot \boldsymbol{\eta}, \nabla \mathbf{v} \cdot \boldsymbol{\eta}) \cdot \mathbf{n}] = 0. \quad (2)$$

Several others jump conditions can be derived from continuity properties across the interface.

- As the fluids are viscous, the velocity field is continuous across the interface, which can be expressed as

$$[\mathbf{u}] = 0, \quad (3)$$

$$[\mathbf{v}] = 0. \quad (4)$$

- The material derivative of (3)-(4) is zero, therefore

$$0 = \frac{\partial [\mathbf{u}]}{\partial t} + (\mathbf{u} \cdot \nabla)[\mathbf{u}] = \left[-\frac{\nabla p}{\rho} + \frac{\nabla \cdot \boldsymbol{\tau}}{\rho} + \mathbf{g} \right],$$

which leads to

$$\left[\frac{\nabla p}{\rho} \right] = \left[\frac{\nabla \cdot \boldsymbol{\tau}}{\rho} \right]. \quad (5)$$

- The jump condition for the pressure p can be simplified. We differentiate in the tangential direction the jump on the velocity:

$$\left[\frac{\partial \mathbf{u}}{\partial \boldsymbol{\eta}} \right] = 0,$$

$$\left[\frac{\partial \mathbf{v}}{\partial \boldsymbol{\eta}} \right] = 0.$$

Moreover, because the velocity is divergent-free on each side of the interface,

$$0 = [\nabla \cdot \mathbf{u}] = \left[(\nabla \mathbf{u} \cdot \mathbf{n}, \nabla \mathbf{v} \cdot \mathbf{n}) \cdot \mathbf{n} + (\nabla \mathbf{u} \cdot \boldsymbol{\eta}, \nabla \mathbf{v} \cdot \boldsymbol{\eta}) \cdot \boldsymbol{\eta} \right]$$

Combining the two last relationships, we obtain

$$[(\nabla \mathbf{u} \cdot \mathbf{n}, \nabla \mathbf{v} \cdot \mathbf{n}) \cdot \mathbf{n}] = 0.$$

Consequently

$$[p] = 2[\mu](\nabla \mathbf{u} \cdot \mathbf{n}, \nabla \mathbf{v} \cdot \mathbf{n}) \cdot \mathbf{n} + \sigma \kappa. \quad (6)$$

We will use this equation to compute the pressure jump at the interface.

Moving interface In the case of a moving interface as it is usually the case for bifluid problem, the flow density and viscosity are updated with ϕ tracking the interface thanks to the transport equation

$$\phi_t + \hat{\mathbf{u}} \cdot \nabla \phi = 0, \quad (7)$$

where the velocity fields $\hat{\mathbf{u}}$ coincides with the flow velocity fields \mathbf{u} on the interface Γ . Different choices for the value of $\hat{\mathbf{u}}$ in Ω^+ and Ω^- can be *a priori* used. A natural choice is to consider $\hat{\mathbf{u}} = \mathbf{u}$ in the whole domain. Another choice is the extension velocity introduced in [1].

3 Numerical method outside the discontinuities

For simplicity reason the computational domain is discretized on a uniform Cartesian grid with a grid spacing $\Delta x = \Delta y = h$, but the following approach stands for non uniform Cartesian meshes. The points on the Cartesian grid are named with indices such as $M_{i,j} = (x_i, y_j)$. We denote by u_{ij} the approximation of u at the point (x_i, y_j) . The narrow band is the set of points having at least one neighbor on the other side of the interface. An accurate computation of the interface is thus necessary.

3.1 Interface computation

In order to provide an accurate discretization in the vicinity of the interface we need some geometric information about the interface, and thus a good choice for ϕ is necessary. This information can be provided by the level set method, introduced by Osher and Sethian [20]. We refer the interested reader to [25], [26] and [19] for recent reviews of this method. A useful choice for the level set function is the signed distance function to the interface:

$$\phi(x) = \begin{cases} \text{dist}_\Gamma(x) & \text{if } x \in \Omega^+, \\ -\text{dist}_\Gamma(x) & \text{if } x \in \Omega^-, \\ 0 & \text{if } x \in \Gamma. \end{cases} \quad (8)$$

The zero isoline thus represents implicitly the interface Γ immersed in the computational domain.

We assume that the interface is smooth enough, so that the derivatives of the level-set function in the vicinity of the interface are well-defined. A useful property of the level set function is

$$\mathbf{n}(x) = \frac{\nabla \phi(x)}{|\nabla \phi(x)|}, \quad (9)$$

where $\mathbf{n}(x)$ is the outward normal vector of the isoline of ϕ passing on x . This allows us to compute the values of the normal to the interface. The curvature of the interface can be computed with the formula

$$\kappa = \nabla \cdot \mathbf{n}. \quad (10)$$

In what follows, the level-set function is advected by the fluid velocity \mathbf{u} in the whole domain

$$\phi_t + \mathbf{u} \cdot \nabla \phi = 0.$$

The computation of the level-set function should be performed very accurately when one deals with moving interfaces, Indeed, as the level-set method is not intrinsically conservative, a lack of accuracy in the computation of the level set evolution results often in a substantial loss of mass for one of the fluids. It can also increase the problem of transfer of momentum between both fluids and generate spurious velocity oscillations. Moreover, if ones wants to compute the curvature of the interface from (10), the

level-set function needs to be accurate enough (at least third-order) so that the finite difference formulas used to discretize (10) are consistent.

Unfortunately, the property of the signed distance function is usually lost when the interface evolves with the flow velocity. The norm of the level set gradient can be far from unity. These gradients variations of the level-set are harmful to the accuracy of the numerical evaluation of the normal to the interface and the curvature. To circumvent the problem, Sussman et al. [17] introduced a reinitialization algorithm to recover the signed distance function through the resolution of the eikonal equation

$$|\nabla\phi| = 1.$$

Several methods have been developed over the years to perform this reinitialization step, either by using a relaxation method and searching a stationary solution to a time dependent Hamilton-Jacobi equation [17] or using a Gauss-Seidel based method as in fast marching methods [27, 22] or fast sweeping methods [33].

The usual level set strategy for an evolving interface is thus the following:

- Use a transport equation to update ϕ .
- From time to time, reinitialize ϕ with the signed distance function.

One of the most widespread option in the literature if ones want a high-order result is to use a fifth-order WENO scheme [28], coupled with a RK3 scheme in time, for the advection of the level-set and for the redistanciation step through a relaxation equation, because it provides a high-order yet stable resolution. But, although the reinitialization procedure performed with such a numerical scheme may improve mass conservation, it also introduces some error by slightly moving the interface as noticed in [24]. For this reason, Russo and Smereka [24] introduced a subcell fix taking into account the interface location in the redistanciation procedure. This technique was extended to a higher order accuracy in [8] through the use of third-order ENO schemes near the interface.

Usually, this redistanciation steps are performed uniformly every each n iterations. A recent study [15] proposes a strategy to sample the reinitialization steps based on the interface deformation criteria. This should also be coupled with the high-order decentered redistanciation scheme of [8] near the interface.

In what follows, we will however use the classical option of the fifth-order WENO scheme for the spatial discretization, because it is the technique most commonly used in the literature in this context, so as to be able to distinguish the effects of the new scheme from the effects of the redistanciation technique.

3.2 Flow computation

We use a classical first-order non-incremental projection method [4, 32], *i.e.* the guess value for the pressure in the prediction step is zero. This choice avoids instability issues due to the discontinuous pressure values when the interface moves, and avoids to wonder which jump conditions across the interface should be satisfied by the pressure correction. Indeed, these jump conditions for the pressure correction are not based on physical considerations, but only on numerical ones, and it would not be straightforward to find out what they should exactly be. We thus compute successively:

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -(\mathbf{u}^n \cdot \nabla)\mathbf{u}^n + \frac{1}{\rho}(\nabla \cdot \boldsymbol{\tau})^n + \mathbf{g} \quad (\text{prediction step}), \quad (11)$$

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^*}{\Delta t} = -\frac{\nabla p}{\rho} \quad (\text{correction step}). \quad (12)$$

The convective terms are computed with a fifth-order WENO scheme, and the viscous terms with an explicit second-order centered finite-difference scheme. The time integration is performed with a first-order explicit Euler scheme.

The pressure is computed through the resolution of a Poisson equation in order to enforce the divergence-free condition. At each grid point outside the narrow band around the interface, the following relationship is satisfied:

$$\nabla \cdot \left(\frac{1}{\rho} \nabla p \right) = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t}. \quad (13)$$

The boundary conditions for the equation on the pressure are of Neumann type:

$$\frac{\nabla p}{\rho} = \frac{\mathbf{u}^b - \mathbf{u}^*}{\Delta t}, \quad (14)$$

with \mathbf{u}^b the value of the velocity that we want to impose on the boundaries. We will provide additional details about the jump conditions that have to be satisfied across the interface for this problem in subsection 4.3.

The overall algorithm is the following:

1. Prediction: evaluate convective and diffusive fluxes and compute \mathbf{u}^* ,
2. Interface evolution: convect the level-set with velocity \mathbf{u} and re-initialize if necessary,
3. Construction and resolution of the linear system for the pressure,
4. Correction step: velocity updated with pressure gradient.

We compute at each iteration an adaptive time step taking into account the restrictions due to convection, viscosity, surface tension and gravity.

The convective time step restriction is given by

$$\Delta t \left(\frac{|u|_{\max}}{\Delta x} + \frac{|v|_{\max}}{\Delta y} \right) \leq 1. \quad (15)$$

with $|u|_{\max}$ and $|v|_{\max}$ the maximum magnitudes of the horizontal and vertical velocities. The viscous time step restriction is given by

$$\Delta t \left(\max\left(\frac{\mu^-}{\rho^-}, \frac{\mu^+}{\rho^+}\right) \left(\frac{2}{\Delta x^2} + \frac{2}{\Delta y^2} \right) \right) \leq 1 \quad (16)$$

The time step restriction associated with the surface tension is similar to the one in [13] and in [6]:

$$\Delta t \sqrt{\frac{\sigma |\kappa|}{\min(\rho^+, \rho^-) \min(\Delta x^2, \Delta y^2)}} \leq 1. \quad (17)$$

We only apply it for grid points in the narrow band.

4 Specific sharp method for the discontinuities across the interface

The values of the viscosity and the densities are discontinuous across the interface. Therefore, if the numerical scheme described above was applied on the grid points in the narrow band, it would not be consistent any more, and stability problems could occur. More precisely, the approximations of the following terms are not consistent if the stencil of the numerical scheme crosses the interface:

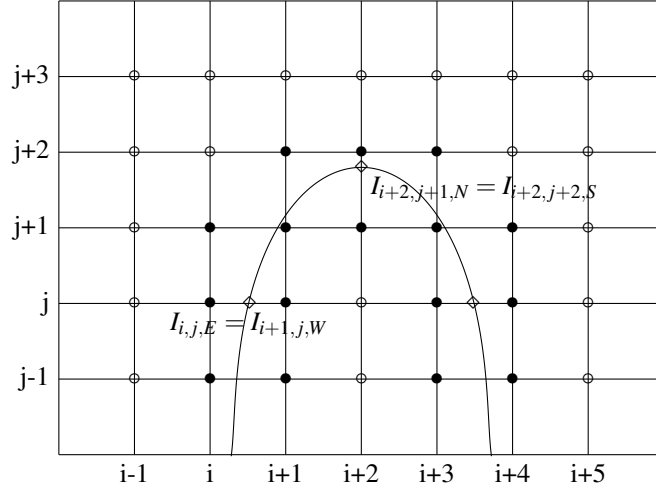


Figure 2: Example of geometrical configuration, with regular grid points (white circles), irregular points (black circles), and interface points (diamonds) with the two possible notations.

- viscous terms,
- divergence of the predicted velocity,
- elliptic operator in the correction step,
- gradient of the pressure.

We have to devise a specific treatment for the points near the interface, for each of these steps. The computation of the convective terms is not mentioned in the above enumeration because it is performed with a fifth order WENO scheme, which provides automatically spatial adaptivity. Therefore, we assume that the gradients computed with the WENO scheme are decentered near the interface, and consequently, consistent. Moreover, the level-set function is classically evolved with such a scheme, because it is crucial to have a good accuracy in the computation of the interface evolution. Therefore, it seems coherent to have the same numerical scheme for the convection of the interface and the convection of the fluids.

Let us introduce some notations. A grid point is defined to be irregular if at least one of its neighbors is on the other side of the interface, *i.e.* if the sign of ϕ changes between this point and at least one of its neighbors, see Figure 2. All the other points are called regular grid points.

We define the interface point $I_{i,j,E} = (\tilde{x}_{i,j,E}, y_j)$ as the intersection of the interface Γ and the *East* segment $[M_{ij}M_{i+1,j}]$, if it exists. In the same way, if the intersection of the interface and the *West* segment $[M_{i-1,j}M_{ij}]$ exists, then we define the interface point $I_{i,j,W} = (\tilde{x}_{i,j,W}, y_j)$ as this intersection. Similarly, the interface points $I_{i,j,N} = (x_i, \tilde{y}_{i,j,N})$ and $I_{i,j,S} = (x_i, \tilde{y}_{i,j,S})$ are respectively defined as the intersection of the interface and the *North* $[M_{ij}M_{i,j+1}]$ and *South* $[M_{i-1,j}M_{ij}]$ segments. With this notation the same interface point can be described in two different ways

$$I_{i,j,S} = I_{i,j-1,N} \text{ or } I_{i,j,E} = I_{i+1,j,W}.$$

The set of interface points, is denoted Γ_h , see Figure 2 for an illustration. These points are used to impose the jump conditions across the interface in the numerical scheme.

4.1 Viscous terms

For the treatment of the viscous terms, we follow a continuous approach and regularize the quantities used for the computation of the viscous terms. It has been proven in [7] and [11] that this continuous approach provides correct accuracy for high Reynolds numbers flows. It has also been used successfully in [21] and [2]. A sharp approach for the viscous terms could probably improve the accuracy of the simulations, but the price to pay would be a substantially increased complexity of the computations, because the jump conditions for the viscous terms (2) imply derivatives of the velocity components in the normal and tangential directions. Moreover, if one needs to use an implicit treatment of the viscous terms, such a sharp treatment would become more complex to handle.

The viscosity and the inverse of the density are regularized in this step by a discrete convolution [21]:

$$16\tilde{\mu}_{i,j} = 4\mu_{i,j} + 2\mu_{i+1,j} + 2\mu_{i-1,j} + 2\mu_{i,j+1} + 2\mu_{i,j-1} + \mu_{i+1,j+1} + \mu_{i+1,j-1} + \mu_{i-1,j+1} + \mu_{i-1,j-1},$$

$$\frac{16}{\tilde{\rho}_{i,j}} = \frac{4}{\rho_{i,j}} + \frac{2}{\rho_{i+1,j}} + \frac{2}{\rho_{i-1,j}} + \frac{2}{\rho_{i,j+1}} + \frac{2}{\rho_{i,j-1}} + \frac{1}{\rho_{i+1,j+1}} + \frac{1}{\rho_{i+1,j-1}} + \frac{1}{\rho_{i-1,j+1}} + \frac{1}{\rho_{i-1,j-1}}.$$

Then we discretize the viscous terms with a classical second-order centered scheme.

Due to this continuous approach for the viscous terms, the jump condition for the pressure (6) writes

$$[p] = \sigma \kappa. \quad (18)$$

and we avoid the computation of the term $2[\mu](\nabla u \cdot \mathbf{n}, \nabla v \cdot \mathbf{n}) \cdot \mathbf{n}$ across the interface.

4.2 Divergence of predicted velocity

The predicted velocity \mathbf{u}^* obtained after the prediction step (11) is defined only on grid points. We need to compute the divergence of this predicted velocity to solve the elliptic equation (13). However, since the two fluids have different properties across the interface and the derivatives of the velocity are not necessarily continuous, we need to use a decentered stencil on each side of the interface. Consequently, we have to compute two values for \mathbf{u}^* on each interface point, one for each side of the interface. In practice, as jump conditions for \mathbf{u}^* are not available, we perform simply linear extrapolations from the grid values on the interface points. Formally this is equivalent to a standard first-order decentered scheme.

Then, to compute the divergence of \mathbf{u}^* on an irregular grid point $M_{i,j}$, we use a standard five point stencil, see Figure 3. More precisely, we denote u_S the value of the solution on the nearest point to $M_{i,j}$ in the south direction (possibly an interface point I), with coordinates (x_S, y_S) . Similarly, we define u_N , u_W and u_E and the associated coordinates (x_N, y_N) , (x_W, y_W) and (x_E, y_E) . The discretization reads

$$\left(\nabla \cdot \mathbf{u}^* \right)_{i,j} = \frac{u_N^* - u_S^*}{x_N - x_S} + \frac{v_E^* - v_W^*}{y_E - y_W}.$$

4.3 Elliptic problem near the interface

To compute the pressure, according to the jump conditions (1) - (6) presented in section 2, it is necessary to solve an elliptic problem with discontinuous values of the solution and its derivative across the interface:

$$\nabla \cdot \left(\frac{1}{\rho} \nabla p \right) = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t} \text{ in } \Omega_1 \cup \Omega_2,$$

$$[p] = \sigma \kappa + 2[\mu](u_n, v_n) \cdot \mathbf{n} \text{ on } \Gamma,$$

$$\left[\frac{\nabla p}{\rho} \right] = \left[\frac{\nabla \cdot \boldsymbol{\tau}}{\rho} \right] \text{ on } \Gamma.$$

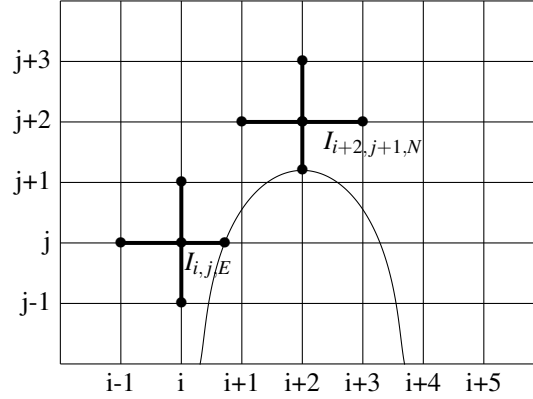


Figure 3: Example of geometrical configuration, with points involved in the discretization of the divergence of the predicted velocity at grid points and gradient of pressure $I_{i,j}$ and $I_{i+2,j+2}$ in black.

Because the viscous terms are handled with a regularization approach, they can not be considered as discontinuous any more. Therefore, the elliptic problem to solve becomes

$$\nabla \cdot \left(\frac{1}{\rho} \nabla p \right) = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t}, \text{ in } \Omega_1 \cup \Omega_2 \quad (19)$$

$$[p] = \sigma \kappa \text{ on } \Gamma, \quad (20)$$

$$\left[\frac{\nabla p}{\rho} \right] = 0 \text{ on } \Gamma, \quad (21)$$

avoiding the computation of the terms $(u_n, v_n) \cdot \mathbf{n}$ and $\left[\frac{\nabla \cdot \boldsymbol{\tau}}{\rho} \right]$. The details of the resolution of this elliptic problems will be provided in section 5.

4.4 Pressure gradient near the interface and correction step

In order to keep a consistent discretization, the gradient of the pressure p computed through the resolution of the elliptic problem in the last subsection, is also computed with an adapted decentered stencil near the interface, see Figure 3. More precisely, with the same notations as before, the discretization reads

$$(\nabla p)_{i,j} = \begin{pmatrix} \frac{p_N - p_S}{x_N - x_S} \\ \frac{p_E - p_W}{y_E - y_W} \end{pmatrix}. \quad (22)$$

5 Numerical resolution of elliptic problems with immersed interfaces

The elliptic problem with discontinuous values across an interface (19) - (21) is solved with the second-order method developed in [5]. The accuracy of this method is based on the use of unknowns located at the interface. These interface unknowns are used to discretize the flux jump conditions and the elliptic operator accurately enough to get a second order convergence in maximum norm. Actually, near the

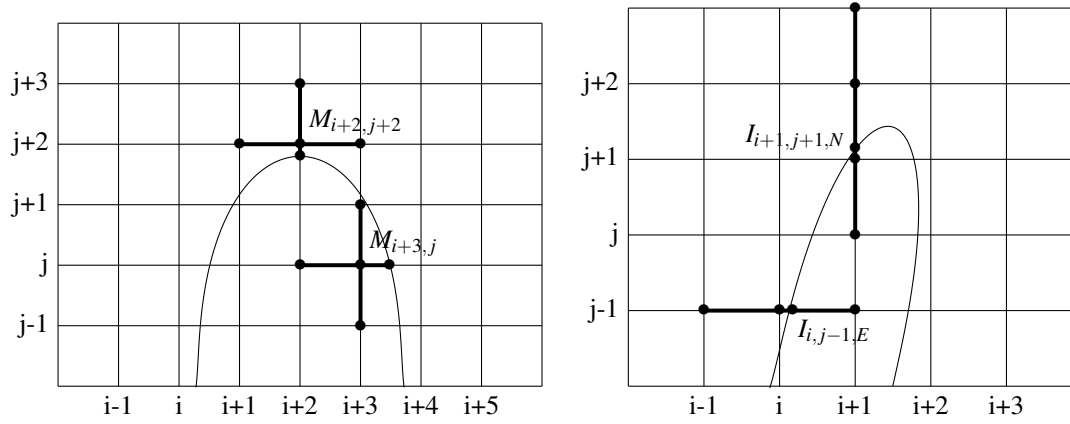


Figure 4: Left: points involved in the discretization of the elliptic operator at grid nodes $M_{i+2,j+2}$ and $M_{i+3,j}$ in black, right: Example of stencils for the discretization of jump conditions. Points involved in the discretization of the x-derivative of the pressure at interface point $I_{i,j-1}$ and y-derivative of the pressure at interface point $I_{i+1,j+1}$ in black. For $I_{i,j-1}$ both derivatives are expressed with second-order accuracy while for $I_{i+1,j+1}$ the left derivative is expressed with second-order and the right derivative with first-order accuracy.

interface, the elliptic operator needs to be discretized with a first order truncation error, while the fluxes have to be discretized with a second-order truncation error. For a visual explanation of the discretization we refer to Figure 4. The advantage of using this method, compared to the reference work of [13] is that the jump conditions in the correction step are solved with second-order accuracy instead of first-order.

5.1 Discrete elliptic operator

We use a standard five point stencil. We use the grid point $M_{i,j}$ and its nearest neighbors in each direction: interface or grid points. More precisely, we denote p_S the value of the solution on the nearest point in the south direction, with coordinates (x_S, y_S) . Similarly, we define p_N , p_W and p_E and the associated coordinates (x_N, y_N) , (x_W, y_W) and (x_E, y_E) . Since the density is piecewise constant, the discretization reads

$$\left(\nabla \cdot \left(\frac{1}{\rho^\pm} \nabla p \right) \right)_{i,j} = \frac{1}{\rho^\pm} \Delta p = \frac{1}{\rho^\pm} \frac{p_N - p_{ij} - p_{ij} - p_S}{x_N - x_i - x_i - x_S} + \frac{1}{\rho^\pm} \frac{p_E - p_{ij} - p_{ij} - p_W}{y_E - y_j - y_j - y_W}. \quad (23)$$

5.2 Discrete flux transmission conditions

At each interface point we create two additional unknowns, called interface unknowns, and denoted by $\tilde{u}_{i,j,\gamma}$ with $\gamma = E, W, N$ or S . The interface unknowns carry the values of the numerical solution on each side of the interface.

Contrarily to [5], we do not have a jump condition on the normal derivative, but on the whole gradient, as expressed in formula (21). It means that to create a numerical system to solve, we have to chose in which direction we want to project this gradient equality. As we use a Cartesian grid, it is easier to discretize the x - and y -derivatives than derivatives in other directions. Consequently, we discretize the

following jump conditions at each interface point $I_{i,j,\gamma}$, with $\gamma = N, S, W, E$.

$$p_{i,j,\gamma}^+ - p_{i,j,\gamma}^- = \sigma \kappa_{i,j,\gamma}, \quad (24)$$

$$\frac{1}{\rho^+} (\partial_x p^+)_{i,j,\gamma} - \frac{1}{\rho^-} (\partial_x p^-)_{i,j,\gamma} = 0 \text{ if } \gamma = E, W. \quad (25)$$

$$\frac{1}{\rho^+} (\partial_y p^+)_{i,j,\gamma} - \frac{1}{\rho^-} (\partial_y p^-)_{i,j,\gamma} = 0 \text{ if } \gamma = N, S. \quad (26)$$

We want the truncation error of the discretization of flux equality (25-26) to be second order accurate, as in [5], in order to solve the problem with a second-order accuracy. A possible configuration of the interface is illustrated in Figure 4. In the x-direction, it is straightforward to compute a second order approximation of the x-derivative with three *a priori* non equidistant points. For example we approximate the flux on the left side of interface point $I_{i,j,E}$, if it exists, with the values of p on the points $M_{i-1,j}$, $M_{i,j}$ and $I_{i,j,E}$ with the formula:

$$(\partial_x p^l)_{i,j,E} \approx \frac{(p_{i-1,j} - \tilde{p}_{i,j,E})(x_i - \tilde{x}_{i,j,E})}{\Delta x(x_{i-1} - \tilde{x}_{i,j,E})} - \frac{(p_{i,j} - \tilde{p}_{i,j,E})(x_{i-1} - \tilde{x}_{i,j,E})}{\Delta x(x_i - \tilde{x}_{i,j,E})}. \quad (27)$$

The right x-derivative $(\partial_x p^r)_{i,j,E}$ is approximated in the same way.

$$(\partial_x p^r)_{i,j,E} \approx -\frac{(p_{i+2,j} - \tilde{p}_{i,j,E})(x_{i+1} - \tilde{x}_{i,j,E})}{\Delta x(x_{i+2} - \tilde{x}_{i,j,E})} + \frac{(p_{i+1,j} - \tilde{p}_{i,j,E})(x_{i+2} - \tilde{x}_{i,j,E})}{\Delta x(x_{i+1} - \tilde{x}_{i,j,E})}. \quad (28)$$

The same discretization is used for the y-derivative. The formulas (27) and (28) are consistent if both grid points involved in the formula, for instance $M_{i-1,j}$ and $M_{i,j}$, belong to the same domain. We thus need that there are at least two adjacent points in each direction belonging to the same domain. If on one side of the interface the two closest grid points aligned with the intersection point do not belong to the same subdomain, then the second-order discretization is not possible anymore. In this case, we use instead for this side of the interface a first order discretization involving only two points: the interface point and the closest grid point on the same side of the interface. Such a case is illustrated on Figure 4. In fact, this first-order discretization is for this special case where the jump in the fluxes is zero, equivalent to the ghost-fluid method [13].

Let us notice that, because we use a dimensional splitting for the jump conditions across the interface, it is quite straightforward to eliminate the interface unknowns from the linear system. We simply inject expressions (27) and (28) in the jump condition (25), and use the resulting equality to express $\tilde{p}_{i,j,E}$ as a function of $p_{i-1,j}$, $p_{i,j}$, $p_{i+1,j}$ and $p_{i+2,j}$. This expression for $\tilde{p}_{i,j,E}$ can then be used in the discretization of the elliptic operator (23).

The local curvature $\kappa_{i,j,\gamma}$ at the interface point $I_{i,j,\gamma}$ is computed in the following way. We first compute on all irregular grid points the value

$$\kappa = \frac{\phi_x^2 \phi_{yy} + \phi_y^2 \phi_{xx} - 2\phi_x \phi_y \phi_{xy}}{(\phi_x^2 + \phi_y^2)^{3/2}}$$

with centered second-order finite-difference formulas. Then we perform a one-dimensional linear interpolation of these values on the interface points.

6 Numerical validation

6.1 Parasitic oscillations

This first test case aims to assess the influence of the interface curvature error on the stability of the numerical scheme. A bubble is located at the center of the computational domain. Due to Laplace law

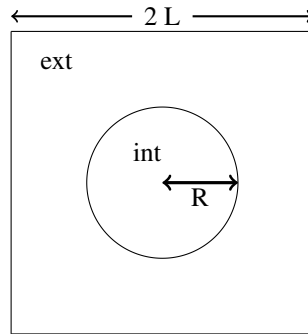


Figure 5: Test case of the static bubble with parasitic oscillations.

and the concavity of the interface, the pressure inside the bubble is larger than the exterior pressure. If the curvature of the interface is computed numerically, the errors due to the numerical approximation in the right-hand side of equation (24) cause small errors in the resolution of the pressure equation, which, in turn, create artificial values of the velocity near the interface, while the velocity should theoretically be zero. These artificial velocities are often called parasitic currents. The amplitude of the parasitic currents is an indication of the stability and the accuracy of the numerical method, and especially of the pressure computational step, because it is the only source of numerical errors.

6.1.1 Comparison with the Ghost Fluid and the CSF methods

We use the same parameters as in [6], where a Ghost-Fluid and a CSF method were implemented. The amplitude of the parasitic currents, compared to the results in [6] reported in Table 1, are presented in Table 2. The L^∞ and L^2 norms are computed over the whole domain Ω . The initial configuration is described in Figure 5. We have performed the computations for two CFL numbers, in order to assess the robustness of our results. As it can be observed, the amplitude of the parasitic currents generated by our new method is several orders of magnitude smaller than those of the CSF method, and significantly lower than those of the Ghost-Fluid method when the grid is refined.

$$\left\{ \begin{array}{l} L = 2 \text{ cm}, \\ R = 1 \text{ cm}, \\ \rho_{int} = 1000 \text{ kg.m}^{-3}, \\ \mu_{int} = 10^{-3} \text{ Pa.s}, \\ \rho_{ext} = 1 \text{ kg.m}^{-3}, \\ \mu_{ext} = 10^{-5} \text{ Pa.s}, \\ \sigma = 0.1 \text{ N.m}^{-1} \end{array} \right. \quad (29)$$

6.1.2 Comparison with a Volume of Fluid method

Now we compare the behaviour of our method to the Volume of Fluid method developed in [30]. The density and viscosity ratio are both chosen to be one for this test-case. We choose $\rho = 1 \text{ kg.m}^{-3}$ and $\mu = 10^{-3} \text{ Pa.s}$. The coefficient σ is chosen so as to obtain an Ohnesorge number $Oh = \frac{\mu}{\sqrt{\sigma\rho D}}$ satisfying $Oh^2 = \frac{1}{12000}$. The maximum velocity is computed for varying grids at non-dimensional time $t^* = \frac{t}{T} =$

N	Ghost Fluid method		CSF	
	L^∞ error	L^2 error	L^∞ error	L^2 error
8	(des.)	(des.)	(des.)	(des.)
16	8.08×10^{-3}	1.88×10^{-3}	3.55×10^{-2}	1.94×10^{-2}
32	3.42×10^{-4}	7.50×10^{-5}	3.12×10^{-2}	1.18×10^{-2}
64	5.13×10^{-5}	7.97×10^{-6}	2.12×10^{-2}	5.44×10^{-3}
128	2.79×10^{-5}	4.74×10^{-6}	6.44×10^{-3}	1.38×10^{-3}

Table 1: Numerical results obtained in [6] for the ghost-fluid method and the CSF method, at time $t = 1$. (des.) indicates that the simulation broke up before the final time.

N	CFL = 0.3				CFL = 0.8			
	L^∞		L^2		L^∞		L^2	
	error	order	error	order	error	order	error	order
8	(des.)	-	(des.)	-	(des.)	-	(des.)	-
16	(des.)	-	(des.)	-	5.21×10^{-3}	-	7.31×10^{-5}	-
32	1.91×10^{-4}	-	3.55×10^{-6}	-	9.26×10^{-5}	5.81	1.42×10^{-6}	5.69
64	1.30×10^{-5}	3.88	1.38×10^{-7}	4.68	1.36×10^{-5}	4.29	1.47×10^{-7}	4.48
128	2.11×10^{-6}	3.25	1.68×10^{-8}	3.86	2.22×10^{-6}	3.73	1.92×10^{-8}	3.96

Table 2: Numerical results for parasitic oscillations, at $t = 1$, (des.) indicates that the simulation broke up before the final time.

250, with $T = \frac{D\mu}{\sigma}$.

$$\left\{ \begin{array}{l} L = 1.25 \text{ m}, \\ R = 1 \text{ m}, \\ \rho_{int} = 1 \text{ kg.m}^{-3}, \\ \mu_{int} = 10^{-3} \text{ Pa.s}, \\ \rho_{ext} = 1 \text{ kg.m}^{-3}, \\ \mu_{ext} = 10^{-3} \text{ Pa.s}, \\ \sigma = 0.00012 \text{ N.m}^{-1} \end{array} \right. \quad (30)$$

The numerical results are presented in Table 3, and suggest a second-order accuracy for our new method. It provides a better accuracy than the Volume of Fluid method for the coarsest grid, but when the grid is refined, the Volume of Fluid method becomes more accurate. However, the fact that our method, which is more simple to implement, can give comparable results to a Volume of Fluid method, is a positive result in itself.

Δx	error L^∞ for [30]	error L^∞ for our method
2.5/16	7.34×10^{-4}	7.48×10^{-5}
2.5/32	4.5×10^{-6}	4.7×10^{-6}
2.5/64	5.5×10^{-8}	1.26×10^{-6}

Table 3: Numerical results for parasitic oscillations at non-dimensional time $t^* = 250$ for [30] and our method.

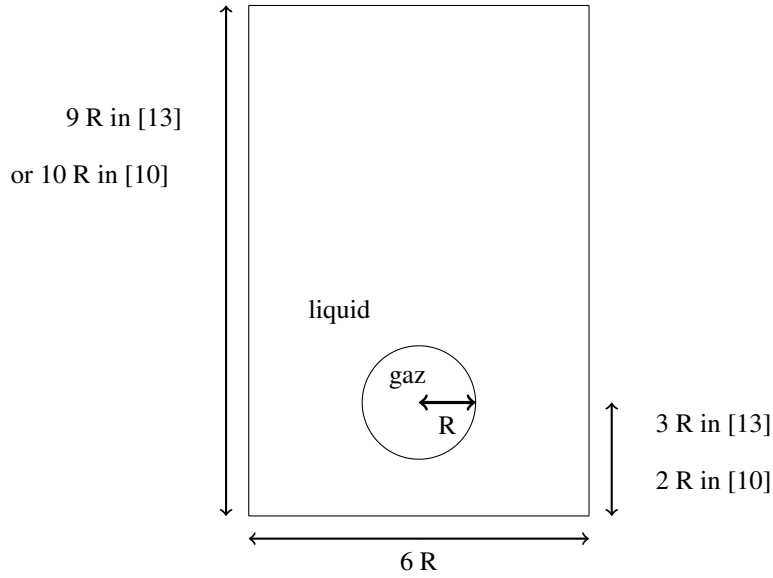


Figure 6: Initial fluid domain for the test case of the rising bubble in water in the references [13] and [10]

6.2 Rising of air bubble in water

We study the evolution of fluid bubbles rising in an heavier fluid, and compare our results to several methods in the literature. The initial configuration is described in Figure 6.

6.2.1 Comparison with the Ghost-Fluid method

We consider air bubbles rising in water, as in the test case proposed for the Ghost-Fluid method in [13]. The value of the physical parameters are

$$\left\{ \begin{array}{l} R = 1/300 m \text{ (small bubble)} \\ R = 1/3m \text{ (large bubble)} \\ \rho_{water} = 1000 \text{ kg/m}^3, \\ \mu_{water} = 1.137 \times 10^{-3} \text{ kg/ms}, \\ \rho_{air} = 1.226 \text{ kg/m}^3, \\ \mu_{air} = 1.78 \times 10^{-5} \text{ kg/ms}, \\ \sigma = 0.0728 \text{ kg/s}^2 \\ g = -9.8m/s^2 \end{array} \right. \quad (31)$$

We consider two cases: a small bubble with $R = 1/300m$ and a large one $R = 1/3m$. In the first case, the surface tension plays an important role in the evolution of the interface because of the small size of the water subdomain. In the second case, the surface tension has less influence, and larger deformations occur. The interface of the small bubble is plotted at times $t = 0, 0.02, 0.035, 0.05$ in Figures 11 and 8. The interface of the large bubble is plotted at times $t = 0, 0.2, 0.35, 0.5$ in Figures 9 and 10. Our numerical results are in good agreement with the ones in [13].

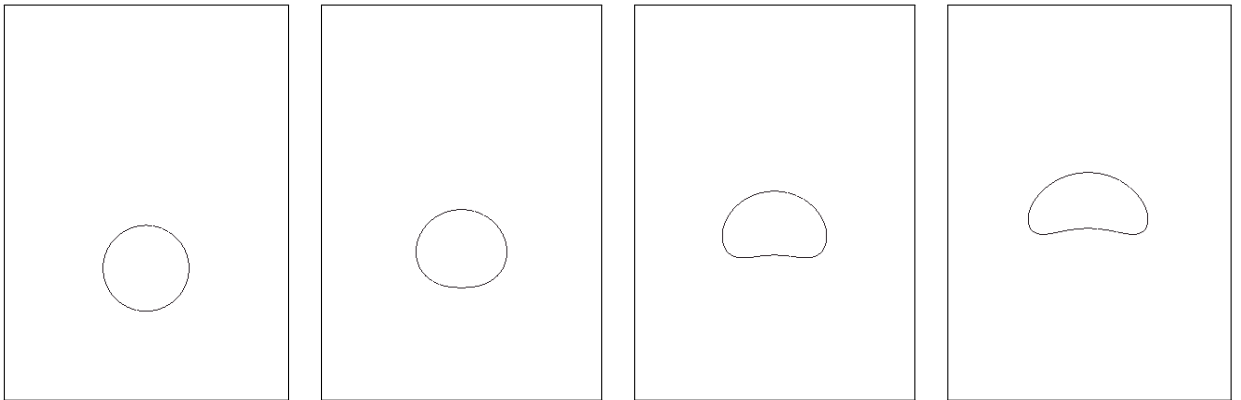


Figure 7: Evolution of the interface for the small bubble test case, resolution 80×120 .

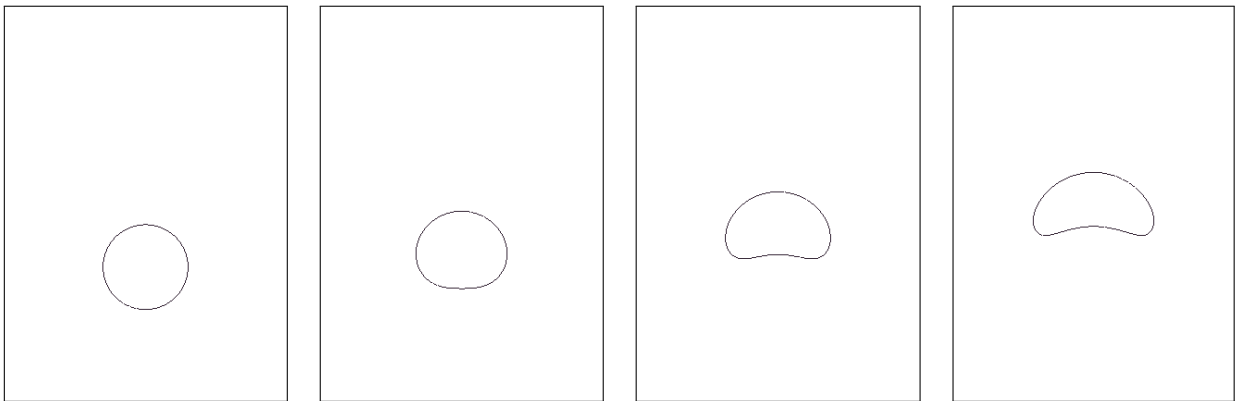


Figure 8: Evolution of the interface for the small bubble test case, resolution 160×240 .

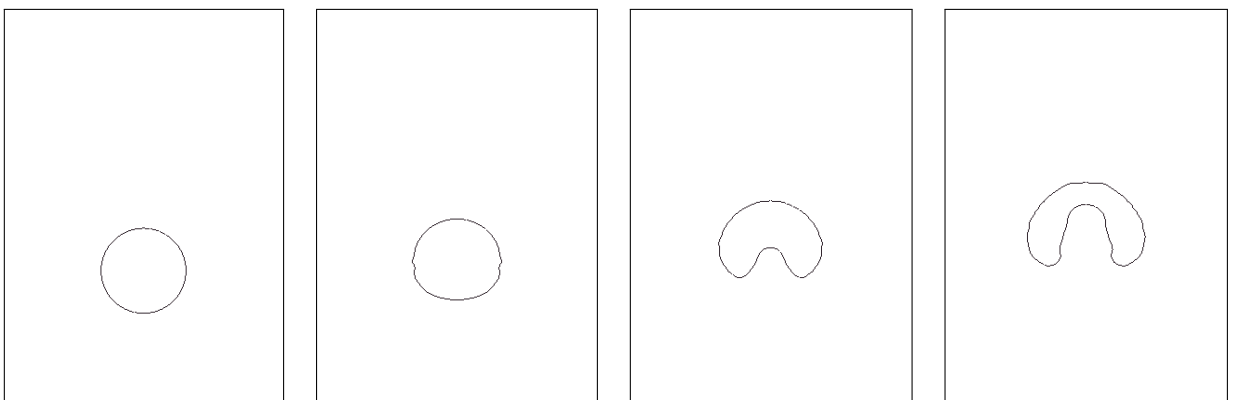


Figure 9: Evolution of the interface for the large bubble test case, resolution 80×120 .

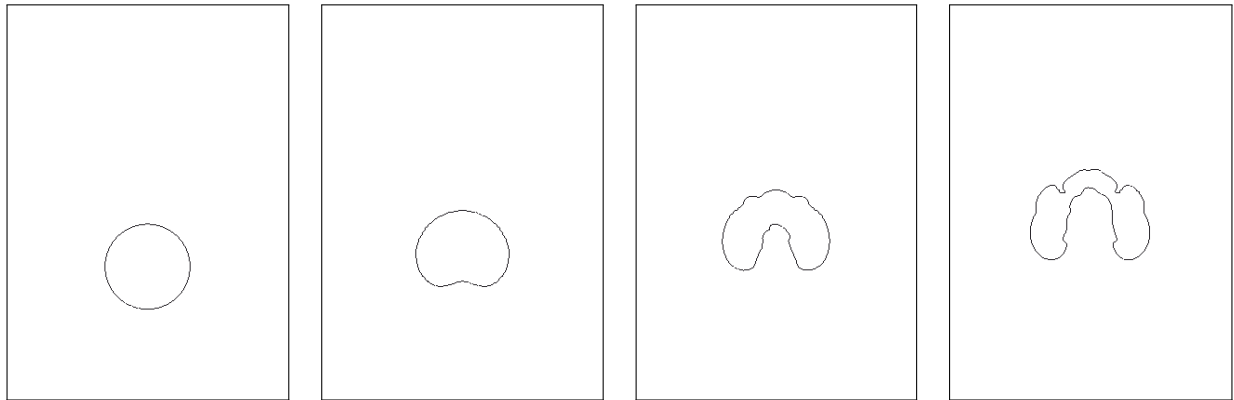


Figure 10: Evolution of the interface for the large bubble test case, resolution 160×240 .

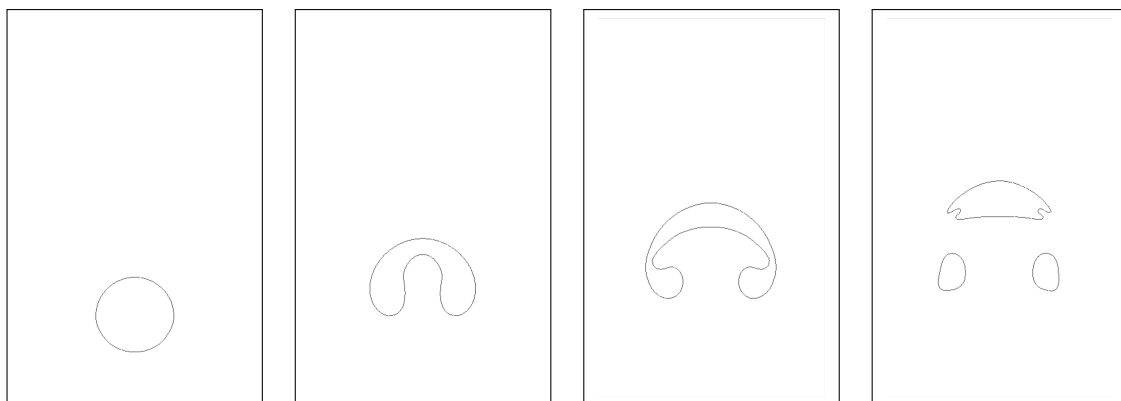


Figure 11: Evolution of the interface for the bubble test case from [10], resolution 120×200 .

6.2.2 Comparison with SPH [10] and the level-set method [17]

This test case is taken from [10], and inspired from a test case presented in [17]. It gives us the opportunity to compare our method to another class of methods, based on the SPH formulation. The initial configuration is described on Figure 6.

The value of the physical parameters are

$$\left\{ \begin{array}{l} R = 0.025 \text{ m} \\ \rho_{\text{water}} = 1000 \text{ kg/m}^3, \\ \mu_{\text{water}} = 1.137 \times 10^{-3} \text{ kg/ms}, \\ \rho_{\text{air}} = 1.226 \text{ kg/m}^3, \\ \mu_{\text{air}} = 1.78 \times 10^{-5} \text{ kg/ms}, \\ \sigma = 0.0728 \text{ kg/s}^2 \\ g = -9.8 \text{ m/s}^2 \end{array} \right. \quad (32)$$

The evolution of the interface is plotted on Figure 11, for 120×200 grid points. We observe that the interface deforms in a way similar to the results in [10].

6.3 Collapse of a water column (dam break)

This test case is studied in [21] and [3], and based on experiments conducted in [18]. The initial configuration is a water column at rest in air. The initial height and width of the column are both 5.715 cm. The domain size is 40 cm \times 10 cm. The physical constants are the same than for the rising bubble in the subsection 6.1.1. For more details, we refer the reader to [21]. We present in Figure 12 the interface evolution at non-dimensional times $T = t\sqrt{g/h} = 0, 1, 2, 3, 4$, with h the initial height of the water column. The computations are performed with 256×64 points.

In Figure 13, we plot the evolution in time of the water front, compared to the experimental results of [18], and to the results obtained for the Ghost-Fluid method and the conservative method of Raessi and Pitsch [21]. We observe that the front propagation is in agreement with the experimental results and the results of the conservative method presented in [21]. It means that, though the method is not strictly conservative, the numerical errors due to momentum transfer across the interface are not large enough to slow down the propagation of the front. It is not the case for instance for the Ghost-Fluid method, as it can be noticed in Figure 13 and has been reported in [21].

7 Conclusion

We have developed a new method on Cartesian grids for the simulation of incompressible fluids with large density ratios. This method is based on a sharp resolution of the pressure correction term, with a second-order Cartesian scheme. This second-order Cartesian scheme, introduced in [5], uses additional unknowns located on the interface to discretize with second-order accuracy the jump conditions across the interface. Moreover, the viscous terms are treated with a regularizing approach, because this regularization permits to eliminate terms in the jump conditions without damaging the accuracy of the results. Numerical results show that the increased accuracy of this new method, compared to state-of-the-art methods on Cartesian grid, leads to more accurate and stable results. In the same time, the resulting numerical scheme remains simple to implement, because it amounts to modifying the stencil of the pressure equation for irregular grid points by adding one additional point. Future works include an extension of the method to three-dimensional problems, possibly with interactions with solids, as for instance in [16]. We also aim to study in details the effects of the redistanciation procedure presented in [15].

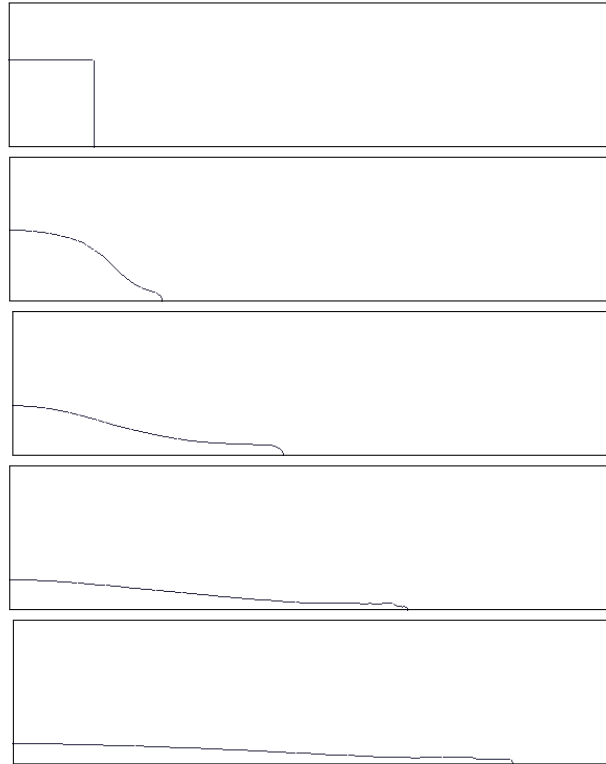


Figure 12: Evolution of the interface for the dam break problem at non-dimensional times $T = t\sqrt{g/h} = 0, 1, 2, 3, 4$.

References

- [1] D. Adalsteinsson and J. A. Sethian. The fast construction of extension velocities in level set methods. *J. Comput. Phys.*, 148:2–22, 1999.
- [2] J. U. Brackbill, D. B. Kothe, and C. Zemach. A continuum method for modeling surface tension. *J. Comput. Phys.*, 100(2):335–354, 1992.
- [3] V. Le Chenadec and H. Pitsch. A monotonicity preserving conservative sharp interface flow solver for high density ratio two-phase flows. *J. Comput. Phys.*, 249:185–203, 2013.
- [4] A.J. Chorin. Numerical solution of the Navier-Stokes equations. *Math. Comp.*, 22:745–762, 1968.
- [5] M. Cisternino and L. Weynans. A parallel second order Cartesian method for elliptic interface problems. *Commun. Comput. Phys.*, 12(5):1562–1587, 2012.
- [6] F. Couderc. *Développement d'un code de calcul pour la simulation d'écoulements de fluides non miscibles: application à la désintégration assistée d'un jet liquide par un courant gazeux*. PhD thesis, ENSAE, Toulouse, 2007.
- [7] O. Desjardins and H. Pitsch. A spectrally refined interface approach for simulating multiphase flows. *J. Comput. Phys.*, 228:1658–1677, 2009.

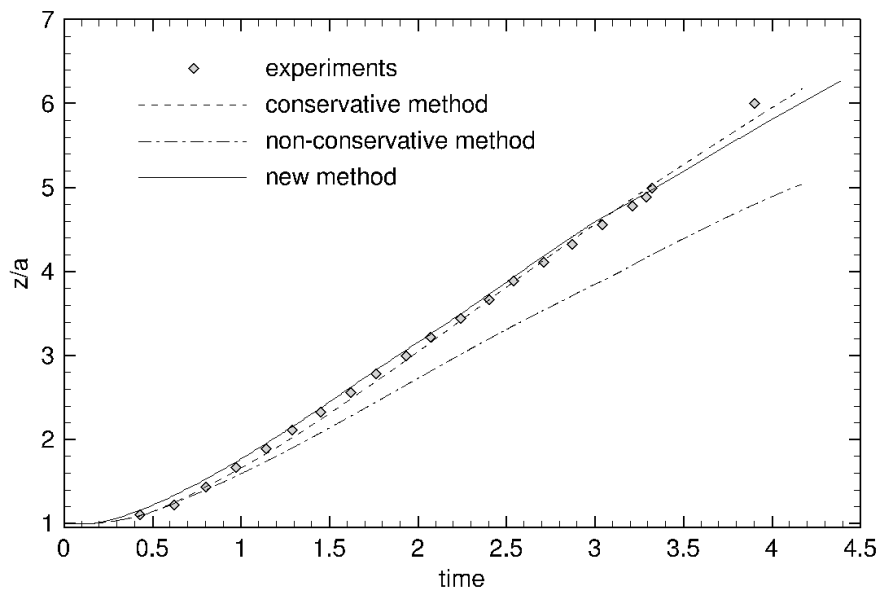


Figure 13: Evolution of the front of propagation: comparison between experimental data and several numerical methods: the Ghost Fluid method (non-conservative method), the conservative method of Raessi and Pitsch and our new method. The dimensionless location of the front $\frac{z}{a}$ is plotted as a function of the dimensionless time $t\sqrt{qh}$.

- [8] A. duChene, C. Min, and F. Gibou. Second-order accurate computation of curvatures in a level set framework using novel high-order reinitialization schemes. *J. Sci. Comput.*, 35:114–131, 2008.
- [9] C. Galusinski and P. Vigneaux. On stability condition for bifluid flows with surface tension: Application to microfluidics. *J. Comput. Phys.*, 227:6140–6164, 2008.
- [10] N. Grenier, M. Antuono, A. Colagrossi, D. Le Touze, and B. Alessandrini. An hamiltonian interface sph formulation for multi-fluid and free-surface flows. *J. Comput. Phys.*, 228:8380–8393, 2009.
- [11] M. Herrmann. The influence of density ratio on the primary atomization of a turbulent jet in cross-flow. *Proc. Combust. Inst.*, 33:2079–2088, 2011.
- [12] X.Y. Hu, B. C. Khoo, N. A. Adams, and F. L. Huang. A conservative interface method for compressible flows. *J. Comput. Phys.*, 219:553–578, 2006.
- [13] Myungjoo Kang, Ronald P. Fedkiw, and Xu-Dong Liu. A boundary condition capturing method for multiphase incompressible flow. *Journal of Scientific Computing*, 15(3):323–360, 2000.
- [14] X.-D. Liu, R. P. Fedkiw, and M. Kang. A boundary capturing method for poisson’s equation on irregular domains. *J. Comput. Phys.*, 160:151–178, 2000.
- [15] F. Luddens, M. Bergmann, and L. Weynans. Enablers for high-order level set methods in fluid mechanics. *Int. J. Numer. Meth. Fluids*, 79:654–675, 2015.
- [16] Bergmann M., Hovnanian J., and Iollo A. An accurate cartesian method for incompressible flows with moving boundaries. *Comm. Comput. Phys.*, 15:1266–1290, 2014.

- [17] M. Smereka M. Sussman and S. Osher. A level-set approach for computing solutions to incompressible two-phase flows. *J. Comput. Phys.*, 114:146–159, 1994.
- [18] J. C. Martin and W. J. Moyce. An experimental study of the collapse of liquid columns on a rigid horizontal plane. *Philos. Trans. R. Soc. London, Ser. A*, 244:312–324, 1952.
- [19] S. Osher and R. Fedkiw. *Level Set Methods and Dynamic Implicit Surfaces*. Springer, 2003.
- [20] S. Osher and J. A. Sethian. Fronts propagating with curvature-dependent speed: Algorithms based on hamiltonâ jacobi formulations. *J. Comput. Phys.*, 79(12), 1988.
- [21] M. Raessi and H. Pitsch. Consistent mass and momentum transport for simulating incompressible interfacial flows with large density ratios using the level set method. *Computers and Fluids*, 63:70–81, 2012.
- [22] E. Rouy and A. Tourin. A viscosity solutions approach to shape-from-shading. *SIAM J. Numer. Anal.*, 29:867–884, 1992.
- [23] M. Rudman. A volume-tracking method for computing incompressible multifluid flows with large density variations. *Int. J. Numer. Meth. Fluids*, 28:357–378, 1998.
- [24] G. Russo and P. Smereka. A remark on computing distance functions. *J. Comput. Phys.*, 163:51–67, 2000.
- [25] J. A. Sethian. *Level Set Methods and Fast Marching Methods*. Cambridge University Press, Cambridge, UK, 1999.
- [26] J. A. Sethian. Evolution, implementation, and application of level set and fast marching methods for advancing fronts. *J. Comput. Phys.*, 169:503–555, 2001.
- [27] J.A. Sethian. A fast marching level set method for monotonically advancing fronts. *Applied Mathematics*, 93:1591–1595, 1996.
- [28] Guang shan Jiang, Chi-Wang Shu, and In L. Efficient implementation of weighted eno schemes. *J. Comput. Phys*, 126:202–228, 1995.
- [29] M. Sussman and E. G. Puckett. A coupled level set and volume of fluid method for computing 3d and axisymmetric incompressible two-phase flows. *J. Comput. Phys.*, 162:301–337, 2000.
- [30] M. Sussman, K. M. Smith, M. Y. Hussaini, M. Ohta, and R. Zhi-Wei. A sharp interface method for incompressible two-phase flows. *J. Comput. Phys.*, 221:469–505, 2007.
- [31] S. Tanguy, T. Menard, and A. Berlemont. A level set method for vaporizing two-phase flows. *J. Comput. Phys.*, 221:837–853, 2007.
- [32] R. Temam. Sur l’approximation de la solution des equations de navier-stokes par la méthode des pas fractionnaires ii. *Archiv. Rat. Mech. Anal.*, 32:377–385, 1969.
- [33] Y.-H., L.-T. Cheng, S. Osher, and H.-K. Zhao. Fast sweeping algorithms for a class of hamilton-jacobi equations. *SIAM J. Numer. Anal.*, 41:673–694, 2003.



**RESEARCH CENTRE
BORDEAUX – SUD-OUEST**

351, Cours de la Libération
Bâtiment A 29
33405 Talence Cedex

Publisher
Inria
Domaine de Voluceau - Rocquencourt
BP 105 - 78153 Le Chesnay Cedex
inria.fr

ISSN 0249-6399