Dear Author,

Please, note that changes made to the HTML content will be added to the article before publication, but are not reflected in this PDF.

Note also that this file should not be used for submitting corrections.

AUTHOR QUERY FORM

Journal: HFF Article Number: 7776	Please e-mail or fax your responses and any corrections to: E-mail: corrections.esch@elsevier.sps.co.in Fax: +31 2048 52799
---	--	---

Dear Author,

Please check your proof carefully and mark all corrections at the appropriate place in the proof (e.g., by using on-screen annotation in the PDF file) or compile them in a separate list. Note: if you opt to annotate the file with software other than Adobe Reader then please also highlight the appropriate place in the PDF file. To ensure fast publication of your paper please return your corrections within 48 hours.

For correction or revision of any artwork, please consult <http://www.elsevier.com/artworkinstructions>.

Any queries or remarks that have arisen during the processing of your manuscript are listed below and highlighted by flags in the proof. Click on the 'Q' link to go to the location in the proof.

Location in article	Query / Remark: click on the Q link to go Please insert your reply or correction at the corresponding line in the proof
<u>Q1</u>	<p>Please confirm that given name(s) and surname(s) have been identified correctly.</p> <div data-bbox="416 1868 981 1970"><p>Please check this box if you have no corrections to make to the PDF file</p><input data-bbox="868 1885 940 1949" type="checkbox"/></div>

Thank you for your assistance.

Highlights

- Two phase flow with free surface is solved by means of two-fluid model (TFM).
 - Fractional Step method and finite volume technique is used to solve TFM.
 - Conservative Level Set method reduces interface sharpening diffusion problem.
 - Cases including high density ratios and high viscosities validate the models.
-

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Simulation of the two-fluid model on incompressible flow with Fractional Step method for both resolved and unresolved scale interfaces

Xiaofei Hou^a, Joaquim Rigola^a, Oriol Lehmkuhl^{a,b}, Assensi Oliva^{a,*}

^a Centre Tecnològic de Transferència de Calor (CTTC), Universitat Politècnica de Catalunya (UPC), ETSEIAT, C. Colom 11, 08222 Terrassa, Barcelona, Spain

^b Termo Fluids S.L., Avinguda Jaquard 97 1-E, 08222 Terrassa, Barcelona, Spain

ARTICLE INFO

Article history:

Received 29 January 2013
Received in revised form 23 October 2014
Accepted 2 November 2014
Available online xxx

Keywords:

Two-fluid model
Free surface flow
Interface sharpening
Fractional Step method

ABSTRACT

In the present paper, the Fractional Step method usually used in single fluid flow is here extended and applied for the two-fluid model resolution using the finite volume discretization. The use of a projection method resolution instead of the usual pressure-correction method for multi-fluid flow, successfully avoids iteration processes. On the other hand, the main weakness of the two fluid model used for simulations of free surface flows, which is the numerical diffusion of the interface, is also solved by means of the conservative Level Set method (interface sharpening) (Strubelj et al., 2009). Moreover, the use of the algorithm proposed has allowed presenting different free-surface cases with or without Level Set implementation even under coarse meshes under a wide range of density ratios. Thus, the numerical results presented, numerically verified, experimentally validated and converged under high density ratios, shows the capability and reliability of this resolution method for both mixed and unmixed flows.

© 2014 Published by Elsevier Inc.

1. Introduction

The multiphase flow has become increasingly important, and has been an active research topic due to its wide variety of application in many natural and industrial fields, ranging from chemical engineering, petro refining to nuclear technology, multiphase transportation, etc.

Based on the interface structure, two-phase flow can be divided in three major groups and several subgroups: Stratified flow (also called free surface flow), mixed or transitional flow and dispersed flow (Yeoh and Tu, 2009). Depending on the specific problem, many models are used. They can be divided into the following categories: Interface-tracking methods, particle tracking methods and multi-fluid methods.

The interface tracking methods, which are most widely used for the simulation of the transients with moving discontinuous interfaces, are a basic two-phase flow model from the standpoint of the Navier–Stokes equations. There are many types of interface tracking methods, which can be classified according to their approaches used to track the interface: Volume of Fraction (VOF) methods, Level Set methods, Lagrangian grid methods (also called surface

adaptive mesh methods) and Lagrangian–Eulerian methods (also called front-tracking methods) (Hirt and Nichols, 1981; Meier et al., 2002; Osher and Fedkiw, 2001). Although interface tracking method can accurately track the interface, keep the interface sharp and conserve the mass; the disadvantage of interface tracking method is critical: it is impossible to track the surface when the chunks of the particular fluid are smaller than the grid cell in a dispersed flow, and the result of the interface tracking methods loses their physical meaning.

The particle-tracking methods are usually Eulerian–Lagrangian methods, where the fluid flow is calculated on a fixed grid, whereas the point particles, being smaller than the grid size, are tracked in a Lagrangian manner. They are often called the discrete-particle model (DPM), discrete-element model (DEM) (Munkejord, 2005) or Marker and Cell method (MAC) (Garrioch, 2000). Although it can track the fluid with the particles, it fails to provide the premise interface. To obtain the exact location of the interface is necessary to greatly increase the number of particles with an expensive CPU time cost.

Based on multi-fluid method, two fluid model treats the phases as inter-penetrating continua and the whole domain is occupied simultaneously by both phases with the ranging volume fraction at each control volume. Each phase has its respective continuity, momentum and energy equations. Cerne and Petelin (2001) and Yan and Che (2010) respectively coupled the interface tracking

* Corresponding author. Tel.: +34 937 398 192; fax: +34 937 398 920.

E-mail address: cttc@cttc.upc.edu (A. Oliva).

URL: <http://www.cttc.upc.edu> (A. Oliva).

method (VOF) and two fluid model to simulate the complex flow where the stratified flow and dispersed flow coexist. The coupling simulation retains the accuracy of the interface tracking schemes for a simple two-phase flow and eliminates the need for the special closure relations of two-fluid model for the same simple flow.

The primary goal of the present paper is focused on the detailed two-dimensional numerical simulation of gas–liquid flow with small or large length scale interface. The governing equations of two-fluid model for the incompressible interfacial flows are discretized while the drag force, the surface tension model and interface sharpening are described. Although the same model was implemented by Strubelj et al. (2009) with pressure-correction algorithm (SIMPLE) (Patankar, 1980), which is a common solution in single fluid and has been widely extended to multiphase flow, the present paper is focused on Fractional Step algorithm (Yanenko, 1971; Chorin, 1968) being different from pressure-correction algorithm. Another objective of the present work is to validate the capability of the solution algorithm in multiphase flows and explore the possibility of two fluid model in free surface with and without interface sharpening method, which is not totally validated by Strubelj et al. (2009). With the proposed algorithm, the validation of two-fluid model applied on dispersed flow and free surface is performed out under different cases with a wide range of density ratio and viscosity ratio: Rayleigh–Taylor instability case with low density ratio, dam break case with high density ratio and high viscosity; while the pressure jump over a droplet interface, an oscillating droplet and a rising bubble are also tested. Finally, sedimentation case is implemented for the dispersed flow.

2. Mathematical model

2.1. Governing equations

The two-fluid model is an average numerical representation for multiphase flow, characterized by two independent velocity fields which specify the motions of each phase. The basic conservative governing equations consist of two continuity equations and two momentum equations including surface tension. They are derived by averaging the original ones for single phase fluid. Detailed derivation of the two-fluid model can be found in the book (Ishii and Hibiki, 2006). Thus, two sets of mass and momentum conservation governing equations of each k phase are defined as:

$$\frac{\partial(\rho_k \alpha_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k \vec{u}_k) = 0 \quad (1)$$

$$\frac{\partial(\rho_k \alpha_k \vec{u}_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k \vec{u}_k \vec{u}_k) = -\alpha_k \nabla p + \nabla \cdot (\mu_k \alpha_k \nabla \vec{u}_k) + \rho_k \alpha_k \vec{g} + \vec{F}_{D,k} + \vec{F}_{S,k} \quad (2)$$

Assuming that the fluid is incompressible, and there is no phase change through the interface, a non-conservative mathematical formulation form of two-fluid model is written below:

$$\frac{\partial(\rho_k \alpha_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k \vec{u}_k) = 0 \quad (3)$$

$$\frac{\partial \vec{u}_k}{\partial t} + \nabla \cdot (\vec{u}_k \vec{u}_k) = -\frac{1}{\rho_k} \nabla p + R(\vec{u}_k) + \frac{\vec{F}_{D,k}}{\alpha_k \rho_k} \quad (4)$$

where $R(\vec{u}_k)$ is defined as

$$R(\vec{u}_k) = -\nabla \cdot (\vec{u}_k \vec{u}_k) + \frac{\mu_k}{\rho_k} \Delta \vec{u}_k + \vec{g} + \frac{\vec{F}_{S,k}}{\rho_k \alpha_k}$$

Both fluids share the same space and pressure having different velocity fields. In that sense, their both continuity equations are added and can be written in the following form

$$\nabla \cdot (\alpha_1 u_1 + \alpha_2 u_2) = 0 \quad (5)$$

2.2. Drag force

Drag force $\vec{F}_D$ describes interfacial forces in momentum equations (also called interfacial friction forces), referred to forces that oppose the relative motion of an object through a fluid (liquid or gas). Since drag force is dependent on the relative velocities between both phase flows, if interface tracking is implemented within two-fluid model, drag force term has to be taken into account. It is important to highlight that the sum of local drag forces of both phases must be zero: $\vec{F}_{D,2} = -\vec{F}_{D,1}$.

Different drag force formulations were employed in many references (Oey et al., 2001; Cerne and Petelin, 2001; Buwa and Ranade, 2002; Yan and Che, 2010; Ishii and Mishima, 1984). The drag force over k phase usually used by many researchers is $F_{D,k} = (\alpha_k \frac{c_D}{2}) \rho_c A_c |\vec{u}_r| \vec{u}_r$ (Ishii and Mishima, 1984), where subscript c represents the continuous phase, ρ_c is the density of continuous phase, A_c is the volumetric interfacial area, $\vec{u}_r$ is the relative velocity and c_D is an empirical coefficient. Other variant of drag force is $F_{D,d} = -F_{D,c} = \frac{3}{4} \alpha_c \alpha_d \rho_c \frac{c_D}{d} |\vec{u}_r| \vec{u}_r$ (Buwa and Ranade, 2002), where subscript d , c represents dispersed and continuous phase respectively, and d is the diameter of dispersed flow instead of A_c although having the same unit.

In the present paper, the drag force equation for dispersed flow is $F_D = \alpha_1 \alpha_2 \rho_m \frac{c_D}{d} |\vec{u}_1 - \vec{u}_2| (\vec{u}_1 - \vec{u}_2)$ (Strubelj and Tiselj, 2011), where mixture density is ρ_m , drag coefficient c_D is usually a function of bubble or droplet, while the modeling of the interfacial length scale is imposed by the physics of the flow and is equal to the fluid equivalent diameter d . For free surface, drag force applied only to free surface flow is an empirical closure correlations (Strubelj and Tiselj, 2011). The term depends on relative velocity $\vec{u}_2 - \vec{u}_1$, mixture density ρ_m , drag coefficient c_D and interfacial length scale d

$$\vec{F}_{D,1} = \alpha_1 \alpha_2 (\vec{u}_2 - \vec{u}_1) \rho_m \frac{c_D}{d} \quad (6)$$

$$\rho_m = \alpha_1 \rho_1 + \alpha_2 \rho_2 \quad (7)$$

where $c_D = 0.44$ and $d = \frac{c_D \Delta t}{100}$ is large enough for equalizing the velocities of both phases at interface.

2.3. Surface force

Surface force $\vec{F}_S$ is calculated as continuum surface force, based on curvature κ , as follows

$$\vec{F}_S = \sigma \kappa \nabla \alpha_1 \quad (8)$$

where σ is surface tension coefficient. The curvature is calculated as follows

$$\kappa = -\nabla \cdot \vec{n}, \quad \vec{n} = \frac{\nabla \alpha_1}{|\nabla \alpha_1|} \quad (9)$$

Finally, surface tension force can be split between two phases occupying the cell, since two momentum equations are solved in the presented system

$$\vec{F}_{S,k} = \beta_k \vec{F}_S \quad (10)$$

where β_k is the averaged factor of k phase. To evaluate β_k value, Bartosiewicz et al. (2008) proposed two models for averaged factor β_k : the first based on mass average $\beta_k = \alpha_k \rho_k / (\alpha_1 \rho_1 + \alpha_2 \rho_2)$, and the second one based on volume average in the cell $\beta_k = \alpha_k$ (the second model is being applied in this paper).

2.4. Interface sharpening algorithm

As free surface is smeared over several cells, and even if continuity equation is solved with the high resolution scheme, the

smearing is still increasing with time. In the present paper conservative Level Set method is applied (Olsson and Kreiss, 2005). The method consists of two steps: in the first step, the continuity equation for the volume fraction Eq. (1) is solved with high resolution scheme; in the second step an equation that acts as an artificial compression is solved

$$\frac{\partial \alpha_1}{\partial \tau} + \nabla \cdot (\alpha_1 (1 - \alpha_1) \vec{n}) = \varepsilon \Delta \alpha_1 \quad (11)$$

where $\vec{n}$ stands for the normal at the interface, and is calculated only once at the beginning of the second step. τ is an artificial time, not equivalent to the actual time t . ε is a small amount of “viscosity”, which is added to avoid discontinuities. Accuracy is assured when

$$\int |a^{n+1} - a^n| < \varepsilon_T \cdot \Delta \tau \quad (12)$$

for a specified total tolerance ε_T .

The volume fraction values obtained through the Eq. (11) may not satisfy the constraint $\alpha_1 + \alpha_2 = 1.0$, due to the numerical error. Afterwards, the new volume fraction can be handled further using an average

$$\alpha_1^{n+1} = \frac{\alpha_1^{n+1}}{\alpha_1^{n+1} + \alpha_2^{n+1}} \quad \alpha_2^{n+1} = \frac{\alpha_2^{n+1}}{\alpha_1^{n+1} + \alpha_2^{n+1}} \quad (13)$$

3. Numerical algorithm

Since each fluid in the two-fluid model has its own set of governing equations, the solution is further complicated by the couplings of different terms (interchange on mass, momentum and energy). The solution algorithm becomes more important and two principal classes of algorithms have been devised. The first class of approach is a coupled or semicoupled time-marching solution strategy (Liles and Reed, 1978; Mahaffy, 1982; Stewart and Wendroff, 1984), which deals principally with fast transients. When the interaction among the phases is very strong or the processes to be simulated have short time scales i.e. nuclear reactor, the governing equations are more tightly coupled. A fully implicit method or semi-implicit approach, which is derived from Implicit Continuous Eulerian (ICE) (Harlow and Amsden, 1971) method, is preferred. Its advantages is robustness, efficiency, generality and simplicity due to solving the velocity, pressure and volume fraction equations simultaneously, while the principal drawback is high storage requirements (Andrea and Greter, 2007).

The second one consists of segregated algorithms derived from the pressure-based schemes widely used in single-phase flow. Among the solution algorithms, pressure-correction technique caters for implicit-type algorithms of steady or unsteady solutions. In the same way, pressure-correction technique is centered on the basic philosophy of effectively coupling between the pressure and the velocity of which the pressure is linked to the velocity via the construction of a pressure field to guarantee the conservation of mass (Yeoh and Tu, 2009). SIMPLE (Patankar, 1980), IPSA (Spalding, 1979; Spalding, 1983) and PISO (Issa, 1986) are common approaches employed for multifluid flow, and have been investigated by many researchers. A common aspect of these pressure-correction procedures is the adoption of pressure correction iteration solution, which is usually most time-consuming during the process, especially in the fine mesh.

Although the widespread information is available on the single-phase solution algorithm, much less information is available on the multifluid algorithm. Many algorithms and techniques developed for single-fluid flow have neither been fully extended nor applied to the simulation of multifluid flow. Fraction step method

(Yanenko, 1971; Chorin, 1968) being a popular method in a single fluid flow, is also a pressure-based segregated approach and can reduce the calculation quantity greatly in single time level by solving the pressure directly. Considering the weakness of the pressure-correction solution, an incompressible Fractional Step method widely employed in single fluid is extended to multifluid field.

3.1. The application of Fractional Step method

The numerical resolution algorithm applied is the classical Fractional Step projection method. The temporal discretization is based on a central difference scheme for the time derivative term in the momentum Eq. (4)

$$\frac{\partial \phi_k}{\partial t} \Big|^{n+1/2} \approx \frac{\phi_k^{n+1} - \phi_k^n}{\Delta t} + O(\Delta t^2) \quad (14)$$

A fully explicit second-order Adams–Bashforth scheme is used for $R(\phi_k)$ without drag force term is

$$R^{n+1/2}(\phi_k) \approx \frac{3}{2} R^n(\phi_k) - \frac{1}{2} R^{n-1}(\phi_k) + O(\Delta t^2, \Delta x^m) \quad (15)$$

And a first-order backward Euler scheme is considered for the pressure-gradient term. Incompressibility constraint is treated implicitly. Thus, the semi-discretized momentum Eq. (4) is computed to obtain an intermediate predictor as

$$\phi_k^* = \phi_k^n + \Delta t \left[\frac{3}{2} R^n(\phi_k) - \frac{1}{2} R^{n-1}(\phi_k) \right] \quad (16)$$

Regarding source Eq. (4) terms, the interfacial forces near the interface in two phase flow are usually large, making the system of equation stiff. Therefore, governing equations become an hyperbolic system with stiff source terms, which have been widely studied in the past years (Monthe, 2001; Tiselj and Horvat, 2002).

A successful numerical method to overcome the previous method limitation is the operator splitting method (OSM), which splits stiff source term from others terms in Eq. (4). These other terms are integrated with explicit scheme and the stiff part (interfacial forces) is integrated with implicit scheme. The system is then analytically solved and a new predictor is calculated as:

$$\phi_1^p = \frac{\phi_1^* (1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_2} \alpha_1) + \phi_2^* (\Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_1} \alpha_2)}{1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_2} \alpha_1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_1} \alpha_2} \quad (17)$$

$$\phi_2^p = \frac{\phi_1^* (\Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_2} \alpha_1) + \phi_2^* (1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_1} \alpha_2)}{1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_2} \alpha_1 + \Delta t \frac{C_D}{d} \frac{\rho_m}{\rho_1} \alpha_2} \quad (18)$$

Defining pseudo-pressure as $\tilde{p} = \Delta t p^{n+1}$, variables at $n+1$ can be calculated as

$$\phi_k^{n+1} = \phi_k^p - \frac{1}{\rho_k} \nabla \tilde{p} \quad (19)$$

In fact, the above momentum equation just holds where its phase exists. In order to make the equation hold in all the domain, both sides of the equation are multiplied by α_k^{n+1}

$$\alpha_k^{n+1} (\tilde{u}_k^{n+1}) = \alpha_k^{n+1} \left(\tilde{u}_k^p - \frac{1}{\rho_k} \Delta t \nabla p^{n+1} \right) \quad (20)$$

The sum of the two fluid equations yields the combined momentum equation

$$(\alpha_1 \phi_1)^{n+1} + (\alpha_2 \phi_2)^{n+1} = (\alpha_1^{n+1} \phi_1^p + \alpha_2^{n+1} \phi_2^p) - \left(\frac{\alpha_1^{n+1}}{\rho_1} + \frac{\alpha_2^{n+1}}{\rho_2} \right) \nabla \tilde{p} \quad (21)$$

Taking the divergence of the above equation yields a Poisson equation for $\tilde{p}$ and substituting the Eqs. (5) and (22) is obtained

$$\nabla \cdot \left(\frac{\alpha_1^{n+1}}{\rho_1} + \frac{\alpha_2^{n+1}}{\rho_2} \right) \nabla \tilde{p} = \nabla \cdot (\alpha_1^{n+1} \phi_1^p + \alpha_2^{n+1} \phi_2^p) \quad (22)$$

The above Poisson equation is solved with SLU solver package developed by Davis (2013). In order to close Eq. (22), the Neumann boundary condition is imposed. Once the Poisson equation is solved, the new velocity can be calculated as follows:

$$\begin{aligned} \tilde{u}_k^{n+1} &= \tilde{u}_k^p - \frac{1}{\rho_k} \tilde{p}^{n+1}, & \text{if } \alpha_k > 0 \\ \tilde{u}_k^{n+1} &= 0, & \text{if } \alpha_k \leq 0 \end{aligned}$$

Since an explicit time integration scheme is applied, the time-step Δt is bounded by the CFL condition (Courant et al., 1928),

$$(\Delta t)_k \left(\frac{|\tilde{u}_k|}{\Delta x_i} \right)_{\max} \leq C_{conv} = 0.35, \quad (\Delta t)_k \left(\frac{|v_k|}{\Delta x_i^2} \right)_{\max} \leq C_{visc} = 0.2$$

$$\Delta t = \min(\Delta t_1, \Delta t_2)$$

3.2. Convective numerical scheme

In governing equations, convection term based on low order scheme usually leads to strongly diffused solutions, particularly for long integration times. As the two fluid is separated by one interface, and the flux is a discontinuous function, numerical diffusion caused will quickly mix the two phases and the interface will lose its sharpness and disappear. It is, therefore, of considerable interest to employ flux limiter. It can avoid the spurious oscillations (wiggles), which would otherwise occur with high order spatial discretization schemes due to shocks, discontinuities or sharp changes in the solution domain.

Using the flux limiter, the flux F at face in convective term is calculated as $F_{ij} = F_{ij}^L + \phi_{ij}(F_{ij}^H - F_{ij}^L)$, where F^H is a high-order flux calculated with Lax-Wendroff scheme (Lax and Wendroff, 1960), F^L is a low-order flux with upwind scheme and ϕ is the flux limiter. Vanleer flux limiter (Leer, 1977) is applied in the work. It is defined as follows:

$$\phi_{ij} = \frac{|r_{ij}| + r_{ij}}{1 + |r_{ij}|} \quad (23)$$

r is smoothness parameter, which is defined by:

$$r_{ij} = \frac{\beta_p - \beta_D}{\beta_U - \beta_p} = \frac{1}{\beta_{i+1,j} - \beta_{ij}} \begin{cases} \beta_{ij} - \beta_{i-1,j} & \text{when } u_{ij} > 0 \\ \beta_{i+2,j} - \beta_{i+1,j} & \text{when } u_{ij} < 0 \end{cases} \quad (24)$$

where variable β stands for variables that is being calculated. The notations ' U ', ' D ' and ' P ' denote respectively the upstream, downstream points and the point being calculated.

3.3. Numerical procedure

The main strategy for solving the system as Fractional Step method is summarized as:

- Step 1. Input relative parameters (geometry, physical properties, boundary condition).
- Step 2. Calculate the time step.
- Step 3. Evaluate the predictor $R(\tilde{u}_k)$ and intermediate predictor with the Eq. (16).
- Step 4. Substituting the intermediate predictor into the Eqs. (17) and (18) to get the final predictor u^p .
- Step 5. Solve Poisson Eq. (22) and the new velocities are obtained.

Step 6. Solve the continuity Eq. (1) and interface sharpening Eq. (11) to obtain the new volume fraction field.

4. Test cases

To validate and explore the capabilities of Fractional Step method in the two-fluid model with and without Level Set method, different multiphase flow problems with small or large length scale are solved, while the results obtained are compared with the existing experimental, analytical and numerical data. Two free surface cases: Rayleigh–Taylor instability and dam break are presented without considering the surface tension, while three different cases: pressure jump over a droplet, an oscillating droplet and a rising bubble are carried out taking into account surface tension. Finally, a dispersed case–sedimentation is simulated considering only gravity effects.

4.1. Rayleigh–Taylor instability

Rayleigh–Taylor instability has attracted the attention of physicists due to its important role in inertially confined nuclear fusion. At the initial state of the Rayleigh–Taylor instability, a horizontal interface separates two fluids of different densities, when higher density fluid is at the top and lower density fluid is at the bottom. As this configuration is unstable, once a small perturbation is introduced, the heavy fluid falls into the light fluid as a spike, and the light fluid rises into the heavy fluid as a bubble.

4.1.1. Initial and boundary conditions

The relative parameters and conditions are exactly the same as in references (Cerne and Petelin, 2001; Strubelj and Tiselj, 2011). All simulations are performed in a two dimensional closed box (Length $L = 1$ m, Height $H = 5.0$ m) containing two immiscible fluids (see Fig. 1). The interface between two fluids has a shape of weak cosine wave

$$\delta = H - \delta \left(\cos \left(\frac{2\pi x}{L} - \pi \right) + 1 \right) \quad 0 \leq x \leq L \quad (25)$$

where $\delta_0 = 0.001$ m. The properties of higher density fluid are density $\rho_1 = 3$ kg/m³, dynamic viscosity $\mu_1 = 0.03$ Pa · s and the properties of lighter density fluid are density $\rho_2 = 1$ kg/m³, dynamic viscosity $\mu_2 = 0.01$ Pa · s. The gravity is considered as $g = -10$ m/s² and is downward vertically.

Fig. 1. Rayleigh–Taylor instability case.

Non-slip boundary condition and the Neumann condition for velocity $\vec{u}$ are respectively prescribed at the top, bottom and side wall, while a symmetry boundary condition for pressure equation ($\vec{n}_{\text{wall}} \cdot \nabla p = 0$) is imposed at all the walls. Although there is no surface tension considered, the wetting angle is taken as $\theta = 90^\circ$ since it is needed by interface sharpening. Results are presented in terms of dimensionless time $\tau = t/\beta$. The deformation and its amplitude time development is exponential

$$y(\tau) = \delta_{\tau=0} e^{\beta\tau} \quad \beta^2 = gkAt \quad At = \frac{\rho_1 - \rho_2}{\rho_1 + \rho_2} \quad (26)$$

where $\delta_{\tau=0} = 0.002$ m. $k = \pi/L$ is the smallest wave number and At is Atwood number.

4.1.2. Results

Rayleigh–Taylor instability has been carried out considering different numerical schemes. Table 1 details 6 different groups considering continuity and momentum equations with or without interface sharpening. Fig. 2(a) shows the amplitude growth in the evolution process of Rayleigh–Taylor instability under different meshes considering scheme f. It can be observed that at the beginning period ($\tau < 0.15$) the numerical results present a slight disagreement with the analytical data. This effect is due to, the volume fraction near interface is always discontinuous or nonlinear, but the linear interpolation is applied to track the greatest sunken point in the simulation. The problem is specially important at the beginning period where the initial disturbance is very thin, and the relative error is significant. The maximum discrepancy between mesh 32×160 and 64×320 is 17% while the maximum discrepancy between mesh 64×320 and 128×640 is 4.5%, which is clearly reduced with mesh density increasing. The amplitude difference between the numerical and analytical data at $\tau = 0.07$ is 0.002 and at $\tau = 0.3$ is 0.0338 on mesh 64×320 .

Fig. 2(b) demonstrates the amplitude growth between the analytical result and simulation data in the evolution process of Rayleigh–Taylor instability at mesh 64×320 . The numerical results show all the schemes present a very similar result. The discrepancy between analytical and numerical data at $\tau = 0.05$ is 0.0014 and at $\tau = 0.3$ is 0.05. In order to further illustrate the influence of the convective term discretization schemes in momentum, continuity equations and interface sharpening, the different discretized schemes with or without interface sharpening are depicted in Fig. 3.

Furthermore, it can be seen from Fig. 3 that the diffusion of scheme-a is significant, while others schemes show good agreement of overcoming diffusion. Both Level Set interface sharpening and Vanleer flux limiter can reduce the diffusion significantly.

4.2. Dam break

Due to the wide application of dam break in engineering, such as designing offshore and coast structure, it has been widely studied in the literature by many different experimental, theoretical and numerical methods (Garrioch, 2000; Abdolmaleki and Morris-Thomas, 2004). This problem was adopted by several researchers as a benchmark test to validate the numerical performance of the proposed formulations for solving two-liquid interfaces or free-surface flows.

Table 1

The detailed numerical schemes being applied.

	Scheme-a	Scheme-b	Scheme-c	Scheme-d	Scheme-e	Scheme-f
Momentum equation	Upwind	Upwind	Upwind	Upwind	Vanleer	Vanleer
Continuity equation	Upwind	Upwind	Vanleer	Vanleer	Vanleer	Vanleer
Interface sharpening	Without	With	Without	With	Without	With

Fig. 2. Rayleigh–Taylor instability. (a) Amplitude of different meshes at scheme f. (b) Amplitude of different schemes on mesh 64×320 .

Cruchaga et al. (2007) performed a set of dam break experiments using a reservoir with shampoo and water for two different initial column aspect ratios with the corresponding numerical predictions. In their simulation, the formulation-edge-tracked interface locator technique (ETILT) is used, while wall friction and turbulence effects are also included. In order to validate the capability of two-fluid model at high density ratio and high viscosity, the same set of dam break is simulated. The length and height of the whole container is $L = 0.42$ m and $H = 0.44$ m respectively (see Fig. 4). The initial width of the column a is 0.114 m while the liquid aspects ratio is defined as

$$A_r = \frac{b}{a} \quad (27)$$

All the simulations were done on the same mesh size as Cruchaga et al. (2007): 100×75 for water and 60×45 for shampoo with and without additional Level Set method (LSM) in order to validate capability of Fractional Step method under high density and viscosity ratios. The upwind scheme is applied in the discretization of momentum equation while Van leer flux limiter is used in continuity equation. The spread of the water along the floor, the height of the water along the left vertical wall, and the height of the water along the right vertical wall are compared. Since the later period of flow is mixed with turbulent flow, only the early stage of result is compared against the experimental data. In reference, the

Fig. 3. Volume fraction field at different instants for different schemes.

simulation with and without wall friction effects were simulated, respectively. In the present paper, **Cruchaga-1** represents **Cruchaga** numerical results without wall friction, **Cruchaga-2** is numerical results with wall friction and dimensionless position is defined $H(L)/a$, **present-with** and without LSM stands for algorithm with and without additional Level Set method, respectively.

4.2.1. Collapse of a water column

The first case is a water column collapse. The fluid properties are: $\rho_1 = 1000 \text{ kg/m}^3$ and $\mu_1 = 0.001 \text{ kg/m s}$ for the water, and $\rho_2 = 1 \text{ kg/m}^3$ and $\mu_2 = 1.983 \cdot 10^{-5} \text{ kg/m s}$ for the air. **Fig. 5**

illustrates the collapsing process of a water column with aspect ratio $A_r = 2$ with interface sharpening method (due to the space limitation, the evolution process without Level Set method is not presented). Since the turbulent flow is involved in the later stage of water collapse, only the first **0.5 s** was simulated. It can be observed that the column begins to move due to the sudden drawing of the baffle. Afterward the water flows along the bottom of the container until reaches the right wall. The water column starts to climb along the right wall because of the existing inertia.

Fig. 6 shows the numerical result performance of the water column collapse against the reference's experimental data and

Fig. 4. Dam break case.

numerical results (Cruchaga et al., 2007). The numerical results obtained with and without Level Set method show good agreement. They are both very similar with the referred experimental and numerical data on the horizontal position with a maximum time advance of 0.02 s. The left vertical position almost coincides with the experimental and their numerical data, with a time delay of 0.03 s compared to the experimental data, which confirms good performance of the present model in the case. The right vertical position is more similar with the experimental result compared to the referred numerical data. The maximum peak value discrepancy between the experimental data and the present numerical result is 3.5%. The mass conservation performs very well in the case with a range of deviation $\pm 1\%$. The good consistency with and without Level Set method shows the capability of the present Fractional Step method in high density ratio.

4.2.2. Collapse of a shampoo column

The collapse of shampoo column is dam break with high density and viscosity ratios. Shampoo properties are $\rho_1 = 1042 \text{ kg/m}^3$ and $\mu_1 = 8 \text{ kg/m} \cdot \text{s}$ and air properties are $\rho_2 = 1 \text{ kg/m}^3$ and $\mu_2 = 1.983 \cdot 10^{-5} \text{ kg/m} \cdot \text{s}$.

Fig. 7 shows the evolution process and predicts the interface behaviour of a shampoo column collapse with aspect ratio $A_r = 2$ with Level Set method (the evolution process without Level Set method is not shown because of lack of space). It can be seen from the evolution of Fig. 7 that there is a tiny bulge at $t = 0.1 \text{ s}$ and $t = 0.2 \text{ s}$, which is not consistent with the experimental phenomenon. One possible reason is that the drag force parameter being applied in the present model maybe not completely suitable for the flow with high viscosity and there is lack of surface tension force. Fig. 8 exhibits the interface positions at different walls during the evolution of shampoo collapse with and without Level Set method. On the horizontal position comparison, there is a time advance of 0.015 s at beginning and a time delay of 0.025 s at the late stage ($t > 0.3 \text{ s}$). The left vertical position shows a good agreement with the numerical data of Cruchaga while all the four results

Fig. 6. Collapse of a water column with aspect ratio $A_r = 2$. (a) Horizontal point. (b) Left vertical point. (c) Right vertical point.

present significant deviations with the experimental data. Different from the horizontal position and right vertical position comparison, the left vertical position with and without Level Set

Fig. 5. Evolution process of a water column with aspect ratio $A_r = 2$.

Fig. 7. Evolution process of a shampoo column with aspect ratio $A_r = 2$.

Fig. 8. Collapse of a shampoo column with aspect ratio $A_r = 2$. (a) Horizontal point. (b) Left vertical point. (c) Right vertical point.

where C_p is a numerical constant characteristic of the quality of the numerical modeling of surface tension forces (a non-dimensional number similar to a capillary number).

A pressure jump over a droplet interface is presented at different meshes, while the results are depicted at 0.2 s when $U_{\max}$ tends to asymptotic value. Fig. 9 presents the pressure cut-lines at $y = 0.00375$ for various levels of grid refinement and as can be seen the pressure approximation become sharper with the finer grids. The pressure jump and velocities induced by numerical error are given in Table 2. The relative error decreases as the mesh increases. Typical C_p is found between 10^{-3} and 10^{-10} . The minimum C_p for VOF-PLIC approach is $5 \cdot 10^{-5}$ and $5 \cdot 10^{-7}$ for the front tracking

method are not highly coincident while they present a discrepancy from 0.3 s to 0.9 s . In the comparison of the right vertical position, a clear distinction between the present data and experimental data is presented with an average error of 0.3 s before 0.7 s . In the simulation, the shampoo was considered a Newtonian fluid, but actually the shampoo is not a pure Newtonian fluid with a variable viscosity, which is a significant aspect resulting in all the inconsistency observed. The good consistency with and without Level Set method at high viscosity ratio are confirmed further through the shampoo collapsing case.

4.3. Pressure jump over a droplet interface

The static drop in equilibrium without gravity has been tested to validate our surface tension force model, which has been studied as a validation case in many literatures (Francois et al., 2006; Seifollahi et al., 2008; Yabe et al., 2001; Vincent and Caltagirone, 2004). The theoretical jump in pressure across drop in this case is given by $\Delta p = \sigma/R$, where σ is surface tension coefficient and R is radius of a droplet.

The computational domain considered is a two-dimensional square cavity having a side length of $L = 7.5 \text{ mm}$. An ethanol droplet is initially located at the center of the domain with a radius $R = 2 \text{ mm}$. The whole domain full of air is in zero gravity and the surface tension coefficient between the ethanol and air is taken to be $\sigma = 0.02361 \text{ N/m}$. The fluid properties are $\rho_1 = 787.88 \text{ kg/m}^3$, $\mu_1 = 1.2 \cdot 10^{-3} \text{ Pa s}$ for ethanol and $\rho_2 = 1.1768 \text{ kg/m}^3$, $\mu_2 = 1.0 \cdot 10^{-5} \text{ Pa s}$ for the air. The real physical properties of air and ethanol are employed in the present paper, which are different from the Tiselj test case (Strubelj et al., 2009).

Initially the velocity and pressure are fixed to zero. The exact pressure difference Δp is 11.8050 Pa . In the computational results that follow, the error in the pressure jump and in the maximum velocity is investigated. The relative pressure jump error is evaluated as

$$\varepsilon(\Delta p) = \frac{|\Delta p_{\text{numerical}} - \Delta p_{\text{analytical}}|}{\Delta p_{\text{analytical}}} \quad (28)$$

where $\Delta p_{\text{simulation}}$ denotes the averaged pressure difference between inside and outside the drop. To avoid considering the transition region, inside means the cell of volume fraction $\alpha \geq 0.999$.

Spurious currents near the interface are best illustrated in the static drop in equilibrium. The reason of its appearance are numerical errors. The order of magnitude of parasitic velocities can be estimated according to the surface tension and dynamic viscosity of the drop (Vincent and Caltagirone, 2004)

$$u_p = \frac{C_p \sigma}{\mu_1} \quad (29)$$

Fig. 9. Pressure cut-line ($y = 0.00375$) for different mesh sizes.

Table 2

Pressure jump over a droplet and numerically induced velocities in the ethanol on different meshes.

	32×32	64×64	128×128
Δp (Pa)	12.32	11.7359	11.7803
Δp error/(%)	4.36	0.59	0.21
C_p	$1.11e-3$	$7.93e-4$	$4.54e-04$

Fig. 10. Spurious velocity at $t = 0.2$ s.

method (Vincent and Caltagirone, 2004), the present parasitic velocities are in a reasonable range. Fig. 10 show the spurious velocity field at 0.2 s, for the finest mesh 128×128 with a reference vector of 8×10^{-3} . It can be observed from the figure that the spurious velocity is concentrated near the interface region, even if the light spurious velocity exists in other regions.

4.4. Oscillating droplet

Oscillating droplet has been employed to illustrate the dynamic case of surface tension force (Francois et al., 2006; Torres and Brackbill, 2000; Stuckrad et al., 1993). The same test case as (Strubelj et al., 2009; Vincent and Caltagirone, 2004)

Fig. 11. Performance of a drop. (a) North point of ellipse drop. (b) North point of square drop.

Table 3

Oscillation period of droplet in simulation with different numbers cells and time steps.

Mode ($n = 2$)	Analytical	64×64	128×128
τ (s)	1.5888	1.565	1.5978
τ error (%)	–	–1.498	0.057
Mode ($n = 4$)	Analytical	64×64	128×128
τ (s)	0.5024	0.5147	0.515
τ error (%)	–	2.45	2.508

was considered. The ethanol ellipsoid drop or square drop is located in a computational domain of square cavity full of air with zero gravity. The size of the domain is $75 \text{ mm} \times 75 \text{ mm}$. The shape of ellipse drop is given by the equation $(x/0.02856)^2 + (y/0.01782)^2 = 1$. The side length of square drop is 40 mm. The fluid properties are the same as the previous test case. The initial velocity field is set to zero in the whole domain and the Neumann boundary condition was imposed. The theoretical oscillation period is obtained by

$$\omega_0^2 = \frac{(n^3 - n)\sigma}{(\rho_1 + \rho_2)R^3}, \quad \tau_0 = \frac{2\pi}{\omega_0} \quad (30)$$

where ω_0 is oscillation frequency and τ_0 is oscillating time period.

Due to the surface tension, the droplet begins to oscillate. A mesh refinement assessment has been carried out. Fig. 11(a) shows the evolution process of north pole of ellipse drop on mesh size 64×64 and 128×128 . It can be observed that the oscillation damping of mesh size 64×64 is larger than the results on mesh

Fig. 12. Initial bubble position.

Fig. 13. performance of rising bubble with lsm. (a) Mass center. (b) Circularity. (c) Rise velocity.

size 128×128 , which is consistent with the previous observation (Strubelj et al., 2009). The observed oscillation damping is caused by the larger numerical viscosity (discretization error in the convective part, which acts as a diffusive term) in the momentum equation. The oscillation period on fine mesh is more accurate than on the coarse mesh, but both are in the range of $\pm 2\%$. The maximum deviation of mass conservation is 0.02% through the whole process.

The simulation of square drop was also done (Fig. 11(b)). It was found that there is maximum oscillation period discrepancy of $\pm 3\%$ on different mesh sizes, and the drop hardly oscillated any more in the simulation of initially square droplet after 6 s. The mass conserve exactly during the whole process.

In order to illustrate the exact error among different mesh size and time step, Table 3 is given. At smaller time step on the same mesh, oscillation period agree well with the analytical data.

4.5. Rising bubble

The rise of an air bubble in water by buoyancy forces was proposed by Hysing et al. (2009) as benchmark case to validate the code's capability. Results obtained are compared with Tiselj data (Strubelj et al., 2009) and Hysing data (Hysing et al., 2009). The gas bubble of density $\rho_1 = 100 \text{ kg/m}^3$, viscosity $\mu_1 = 1 \text{ Pa} \cdot \text{s}$ and $R_0 = 25 \text{ cm}$ is initially imposed in the liquid with properties $\rho_2 = 1000 \text{ kg/m}^3$, viscosity $\mu_2 = 10 \text{ Pa} \cdot \text{s}$. The chosen fluids are non-realistic and the purpose is to compare the results obtained with different numerical methods. The gravity in the system is $g = -0.98 \text{ m/s}^2$ and the surface tension between fluids is $\sigma = 24.5 \text{ N/m}$. The corresponding dimensionless numbers are $Re = \rho_1 \sqrt{g} (2R_0)^{3/2} / \mu_1 = 35$ and $Eo = 4\rho_1 g R_0^2 / \sigma = 10$. The initial fluids are both at rest initially. The computational domain for the test problem is chosen to be $[0, 1] \times [0, 2]$. The geometry is shown in Fig. 12(a).

Several parameters were analyzed and compared to the results of Strubelj et al. (2009) and Hysing et al. (2009): the position of bubble mass center, the rise velocity, mass conservation of the bubble and the circularity. The numerical simulation were implemented respectively on mesh size 80×160 , 160×320 and 320×640 . Vanleer flux limiter was applied to both continuity equation and momentum equation. With and without Level Set method algorithm were respectively performed in this case for verifying the capability in this moving situation.

Fig. 13 presents the performance of rising bubble with Level Set method in the present simulation. The numerical result were compared with the Tiselj data and Hysing data (Hysing et al., 2009). Seeing that the curve on different meshes are very close in reference (Strubelj et al., 2009), only the result on mesh size 80×160 is employed as the reference in the case. In Fig. 13(a), the results on different mesh size are almost the same, and their mass center curves coincide within the first 2 s. Afterward the difference among them starts to increase but still is not great. It can be observed from Fig. 13(b) that, the results is closer to the Tiselj data than the Hysing curve, which is obtained with Level Set method, but the result move slowly toward the Hysing curve with the

Fig. 14. Performance of rising bubble without LSM. (a) Mass center. (b) Circularity. (c) Rise velocity.

increasing mesh. Fig. 13(c) presents the rising velocity of the bubble at mass center. The similar phenomenon as Fig. 13(b) appears, but the difference among different mesh sizes is quite small and

Table 4
Parameter comparison of bubble development.

Parameter	Hysing	Tiselj	With Level Set method			Without Level Set method		
			80 × 160	160 × 320	320 × 640	80 × 160	160 × 320	320 × 640
Mass center ($t = 3$ s)	1.0817	1.0679	1.0569	1.064	1.066	1.060	1.061	1.061
Max velocity (m/s)	0.2418	0.2457	0.2398	0.2431	0.2443	0.2392	0.2415	0.2429
Time (s)	0.9214	0.9235	0.9301	0.9301	0.9301	0.9251	0.9201	0.9251
Min circularity (m)	0.9012	0.8876	0.8859	0.8895	0.8916	0.8987	0.8928	0.8907
Time (s)	1.8830	1.8915	1.9001	1.9001	1.8901	1.8501	1.8601	1.8701

Fig. 15. Sedimentation case. (a) Initial state. (b) Final state.

can be neglected. The bubble evolution with time can be seen in Fig. 12(b).

Fig. 14 gives the result of a rising bubble without Level Set method at different grids. It shows the similar performance as with Level Set method on bubble mass center position and bubble circularity, while the pronounced difference between them appears on bubble rise velocity. With Level Set method, the data with a coarse mesh is closer to a Tiselj curve between 1.5 s and 2.5 s, and the result is far from the Tiselj curve, and towards the Hysing curve as the mesh size increases. On the contrary, without the Level Set method, the data in coarse mesh is closer to the Hysing curve between 1.5 s and 2.5 s and the curve goes close to the Tiselj curve as the mesh become finer. Meanwhile, the clear disagreement was observed after 2.5 s, especially in the coarse mesh.

All the results show the good consistency with the Tiselj result. The total mass in all the simulations is conserved very well, and the maximum error is less than 0.1%.

In order to compare the key parameters quantitatively, Table 4 was given. In the table, the position of center mass at $t = 3$ s, the maximum rising velocity and its appearing time, the minimum circularity and its appearing time are listed and compared with the Hysing result and Tiselj result. It can be summarized from the table that the results of the two fluid model with and without Level Set

Fig. 16. Comparison between numerical and analytical results at $t = 0.6$ s. (a) Volume fraction. (b) Liquid velocity.

Fig. 17. Comparison between numerical and analytical results at $t = 1.0$ s. (a) Volume fraction. (b) Liquid velocity.

method using Fractional Step method are very similar to the results of Hysing and Tiselj.

4.6. Sedimentation

Sedimentation is a classical dispersed flow case, in which the interface characteristic length scale of fluids are usually smaller than the grid cell. This problem is a simplified physical phenomena proposed by Coquel et al. (1997), based on a separation of air and water by gravity. It consists of a vertical tube of $H = 7.5$ m length and $D = 1$ m diameter, where the gravity is the only source term, any others terms including drag forces are not taken into account. At the initial condition of zero gravity, the tube is filled with stagnant liquid and gas with a homogeneous volume fraction of 0.5 and a uniform pressure of $p_0 = 0.1$ MPa (Fig. 15(a)). The tube is closed at both ends and, thus, the velocities of the two fluids are forced to equal zero at both ends. The Neumann boundary condition is imposed on lateral walls in velocity and volume fraction. At one instant, the gravity is suddenly imposed on the whole system. Due to the gravity effect, the liquid falls downwards to the bottom of the tube, while the gas come up to the top of the tube (Fig. 15(b)). The objective of this problem is to check the capability of the model in the dispersed flow with small scale interface describing countercurrent flow conditions with strong void gradients typical of many situations where phase separation are dominating.

The physical properties for water are: $\rho_1 = 1000$ kg/m³ and $\mu_1 = 0.798 \times 10^{-3}$ Pa · s and for air: $\rho_2 = 1$ kg/m³ and $\mu_1 = 1.983 \times 10^{-5}$ Pa · s. In order to check the mesh independence, the simulations are respectively implemented using 5×75 , 10×150 and

20×300 control volumes. Since it is a dispersed flow, interface sharpening equation in step 6 (Section 3.3) is not implemented in the case. Vanleer flux limiter is applied on continuity and momentum equations, and the obtained results are compared with the analytical solution from literature (Evje and Flåtten, 2003).

The liquid volume fraction distribution along the vertical height at different times is shown in Figs. 16 and 17. The comparisons show that the present numerical results are in good agreement with the analytical ones. Separation point between mixture phase and liquid phase at $t = 0.6$ s (Fig. 16(a)) presents a slight delay in comparison with the analytical prediction with a maximum deviation of 0.1 m. The differences measured by 1-norm $\|E\| = \sum_j |\alpha_j - \alpha_j^{analytical}|$ are 0.2466, 0.1458, 0.0906 at $t = 0.6$ s for three different refined meshes respectively, and 0.1997, 0.1001, 0.0500 at $t = 1$ s for the same mesh sizes. The slight delay of separation point between gas and liquid phase still exists with the same discrepancy of 0.1 m (Fig. 17(a)).

5. Conclusions

In the present work, the two-fluid model has been solved by means of the explicit Fractional Step method avoiding the problem of implicit formulation under pressure-based iteration method. Surface tension is also solved, while diffusion effects have been reduced, considering high resolution schemes and implementing interface sharpening method. Verification tests of mesh density and numerical schemes are carried out in order to confirm the influence of discretization scheme.

The capability of the two-fluid model with Fractional Step method in flow with small or large scale length interface has been

assured by means of different numerical and experimental cases with and without interface sharpening method. In order to highlight the capability of the two fluid model in flow with large scale length, Rayleigh–Taylor instability, dam break, droplet and rising bubble have respectively been implemented. The results obtained are all consistent with the referred data. The case of Rayleigh–Taylor instability illustrates clearly the advantages of Vanleer flux limiter and interface sharpening on controlling the diffusion. Dam break cases shows the good performance in the flow with high density ratio and high viscosity. Through the simulation of pressure jump over droplet interface, oscillating droplet and rising bubble, the capability of the two-fluid model where surface tension dominated was validated. The capability of the two-fluid model in the flow with small scale length interface is presented in the case of sedimentation.

Overall, the present simulations and results conclude that, the two fluid model with small or large characteristic scales can be solved successfully to obtain the accurate results through Fractional Step method algorithm, and it can also be applied to simulate the free surface flow if appropriate formulations and parameters are used. The Fractional Step method is able to solve the large characteristics scale problems without interface sharpening even with coarse meshes, reducing the diffusion and sharp the interface in all situations.

Acknowledgement

This work has been partially financed by the Ministerio de Economía y Competitividad(MINECO), Secretaria de Estado de Investigación, Spain (Ref. ENE 2011-28699)

References

- Abdolmaleki, K., Morris-Thomas, M.T., 2004. Simulation of the Dam Break problem and impact flows using a Navier–Stokes solver. In: 15th Australasian Fluid Mechanics Conference, pp. 13–17.
- Andrea, P., Greta, T., 2007. Computational Methods for Multiphase Flow. Cambridge University Press.
- Bartosiewicz, Y., Lavieville, J., Seynhaeve, J.-M., 2008. A first assessment of the NEPTUNE-CFD code: instabilities in a stratified flow comparison between the VOF method and a two-field approach. *Int. J. Heat Fluid Flow* 29 (2), 460–478.
- Buwa, V.V., Ranade, V.V., 2002. Dynamics of gas–liquid flow in a rectangular bubble column: experiments and single/multi-group CFD simulations. *Chem. Eng. Sci.* 57 (22–23), 4715–4736.
- Cerne, G., Petelin, S., 2001. Coupling of the interface tracking and the two-fluid models for the simulation of incompressible two-phase flow. *J. Comput. Phys.* 171 (2), 776–804(29).
- Chorin, A.J., 1968. Numerical solution of the Navier–Stokes equations. *Math. Comput.* 22 (104), 745–762.
- Coquel, F., Amine, K.E., Godlewski, E., Perthame, B., Rascle, P., 1997. A numerical method using upwind schemes for the resolution of two-phase flows. *J. Comput. Phys.* 136 (2), 272–288.
- Courant, R., Friedrichs, K., Lewy, H., 1928. Über die partiellen differenzengleichungen der mathematischen physik. *Math. Ann.* 100 (1), 32–74.
- Cruchaga, M.A., Celentano, D.J., Tezduyar, T.E., 2007. Collapse of a liquid column: numerical simulation and experimental validation. *Comput. Mech.* 39 (4), 453–476.
- Davis, T.A., 2013. Suitesparse <<http://www.cise.ufl.edu/research/sparse/SuiteSparse/>>.
- Evje, S., Flåtten, T., 2003. Hybrid flux-splitting schemes for a common two-fluid model. *J. Comput. Phys.* 192 (1), 175–210.
- Francois, M.M., Cummins, S.J., Dendy, E.D., Kothe, D.B., Sicilian, J.M., Williams, M.W., 2006. A balanced-force algorithm for continuous and sharp interfacial surface

- tension models within a volume tracking framework. *J. Comput. Phys.* 213 (1), 141–173.
- Garrioch, S.H., 2000. A Volume Tracking Method for the Simulation of Two-Fluid Flows. Ph.D. thesis, McGill University.
- Harlow, F.H., Amsden, A.A., 1971. A numerical fluid dynamics calculation method for all flow speeds. *J. Comput. Phys.* 8 (2), 197–213.
- Hirt, R., Nichols, B., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* 39 (1), 201–225.
- Hysing, S., Turek, S., Kuzmin, D., Parolini, N., Burman, E., Ganesan, S., Tobiska, L., 2009. Quantitative benchmark computations of two-dimensional bubble dynamics. *Int. J. Numer. Methods Fluids* 60 (11), 1259–1288.
- Ishii, M., Hibiki, T., 2006. *Thermo-Fluid Dynamics of Two-Phase Flow*. Springer, New York/Berlin.
- Ishii, M., Mishima, K., 1984. Two-fluid model and hydrodynamic constitutive relations. *Nucl. Eng. Des.* 82 (2–3), 107–126.
- Issa, R., 1986. Solution of the implicitly discretised fluid flow equations by operator-splitting. *J. Comput. Phys.* 62 (1), 40–65.
- Lax, P., Wendroff, B., 1960. Systems of conservation laws. *Commun. Pure Appl. Math.* 13 (2).
- Leer, B.V., 1977. Towards the ultimate conservative difference scheme. IV. A new approach to numerical convection. *J. Comput. Phys.* 23 (3), 276–299.
- Liles, D., Reed, W., 1978. A semi-implicit method for two-phase fluid dynamics. *J. Comput. Phys.* 26 (3), 390–407.
- Mahaffy, J.H., 1982. A stability-enhancing two-step method for fluid flow calculations. *J. Comput. Phys.* 46 (3), 329–341.
- Meier, M., Yadigaroglu, G., Smith, B.L., 2002. A novel technique for including surface tension in PLIC-VOF methods. *Euro. J. Mech.-B/Fluids* 21 (1), 61–73.
- Monthe, L.A., 2001. A study of splitting scheme for hyperbolic conservation laws with source terms. *J. Comput. Appl. Math.* 137 (1), 1–12.
- Munkejord, S.T., 2005. Analysis of the Two-Fluid Model and the Drift-Flux Model for Numerical Calculation of Two-Phase Flow. Ph.D. thesis, Norwegian University of Science and Technology.
- Oey, R., Mudde, R., Portela, L., van den Akker, H., 2001. Simulation of a slurry airlift using a two-fluid model. *Chem. Eng. Sci.* 56 (2), 673–681.
- Olsson, E., Kreiss, G., 2005. A conservative level set method for two phase flow. *J. Comput. Phys.* 210 (1), 225–246.
- Osher, S., Fedkiw, R.P., 2001. Level set method: an overview and some recent results. *J. Comput. Phys.* 169 (2), 463–502.
- Patankar, S.V., 1980. *Numerical Heat Transfer and Fluid Flow*. Hemisphere.
- Seifollahi, M., Shirani, E., Ashgriz, N., 2008. An improved method for calculation of interface pressure force in PLIC-VOF methods. *Euro. J. Mech.-B/Fluids* 27 (1), 1–23.
- Spalding, D.B., 1979. Calculation of Two-Dimensional Two-Phase Flows. Hemisphere.
- Spalding, D.B., 1983. Developments in the ipsa procedure for numerical computation of multiphase flow phenomena. In: Shi, T.M. (Ed.), *Proceedings of the Second National Symposium on Numerical Methodologies in Heat Transfer*. Hemisphere, pp. 421–436.
- Stewart, H.B., Wendroff, B., 1984. Two-phase flow: models and methods. *J. Comput. Phys.* 56 (3), 363–409.
- Strubelj, L., Tiselj, I., 2011. Two-fluid model with interface sharpening. *Int. J. Numer. Methods Eng.* 85 (5), 575–590.
- Strubelj, L., Tiselj, I., Mavko, B., 2009. Simulations of free surface flows with implementation of surface tension and interface sharpening in the two-fluid model. *Int. J. Heat Fluid Flow* 30 (4), 741–750.
- Stuckrad, B., Hiller, W.J., Kowalewski, T.A., 1993. Measurement of dynamic surface tension by the oscillating droplet method. *Exp. Fluids* 15 (4–5), 332–340.
- Tiselj, I., Horvat, A., 2002. Accuracy of the operator splitting technique for two-phase flow with stiff source terms. In: *Proceedings of the 2002 ASME Joint US-European Fluids Engineering Conference*. Montreal, Que, Canada 1, pp. 1165–1171.
- Torres, D., Brackbill, J., 2000. The point-set method: front-tracking without connectivity. *J. Comput. Phys.* 165 (2), 620–644.
- Vincent, S., Caltagirone, J.P., 2004. Test-case no 10: parasitic currents induced by surface tension(PC). *Multiphase Sci. Technol.* 16 (1–3), 69–74.
- Yabe, T., Xiao, F., Utsumi, T., 2001. The constrained interpolation profile method for multiphase analysis. *J. Comput. Phys.* 169 (2), 556–593.
- Yan, K., Che, D., 2010. A coupled model for simulation of the gas–liquid two phase flow with complex flow patterns. *Int. J. Multiphase flow* 36 (4), 333–348.
- Yanenko, N.N., 1971. *The Method of Fractional Steps*. Springer-Verlag.
- Yeoh, G.H., Tu, J., 2009. *Computational Techniques for Multiphase Flow*. Butterworth-Heinemann.