1 INTRODUCTION

The knowledge of two-phase flows comportment is of great importance in various process ranging from engineering applications to environmental phenomena. The presence of air bubbles in hydrodynamic systems often reveals many undesirable phenomena as early erosion, loss of efficiency or flow irregularities. Theoritical study of two-phase flow are carried out to analyse the interface behaviour of one or more air bubbles moving in a liquid. The dynamic and deformation of the liquid-air interface is commonly analysed by Volume Of Fluid (VOF) methods. These methods are widely used for two-phase flow simulations and ensure good agreement between numerical and experimental datas (Chen et al., 1996, Chen et al., 1998). The use of a colour function F allows to localize liquid-air interface by the spacial variation of its values ranging from 0 to 1. The reconstruction of the interface is deduced from F values and geometrical assumptions (Li, 1995, Meier 1999). The numerical Piecewise Line Interface Construction (PLIC-VOF) method is one of the more accurate and less numerically diffusive (Hirt and Nichols, 1979, Noh and Woodward 1976, Rider and Kothe, 1997). The tracking liquid-air interface is associated with an advection equation of which the resolution is carried out with other geometrical assumptions (Meier, 1999, Zaleski, 1999). A robust and accurate algorithm of resolution of the Navier-Stokes equations is necessary to determine the complex topological changes of the interface. The projection algorithm ensures the computationnal efficiency according to the advection equation by using of Courant-Friedrichs-Levy (CFL) condition (Meier, 1999, Rider, 1995). The most widely used approach, based on the CSF (Continuum Surface Force) model developped by Brackbill et al., 1992, allows to consider the surface tension contribution for the interface movement. These methods allow to describe shape evolution of the interface. In this article an improvement of the numerical model used for the simulation of two air bubble coalescence with PLIC-VOF method is presented. A new curvature technique calculation for the CSF model is developed to analyse the surface tension contribution in the Navier-Stokes equations. A new advection algorithm is implemented which ensure more mass conservation and better agreement with Rider & kothe advections tests (Rider et al., 1995) than previous algorithm (Zaleski, 1999, Meier, 1999). Results are presented by a succession of images of the shape evolution of the interface during the coalescence. An experimental study is also presented to validate numerical results.

2 PROBLEM FORMULATION

Let us consider two spherical air-bubbles rising through a quiescent water in a rectangular column (Fig.1). The water and the gas are assumed to be isothermal and incompressible and the coalescence two dimensionnal. We used 2D cartesian coordinates, because our numerical code must be able to analyse, in a quiescent water column, bubbly moving up and bubble's coalescence which can be out of line. The initial shape is defined spherical for the both bubbles and the effect of the initialization on the coalescence phenomenon has not been investigated.

The interface motion is computed by solving the advection equation, where the colour function, commonly noted C or F indicates the volumic fraction of liquid, ranging from 0 to 1 within the numerical domain. The PLIC-VOF method is a tracking interface method which reconstructs the interface by a segment of straight line within each calculation cell. This method is of second order accuracy, less diffusive than previous methods (Noh and Woodward, 1976). Moreover it allows a better mass conservation. The density and viscosity properties are then deduced from the following expressions:

$$\rho = \rho_l F + (1 - F)\,\rho_g \tag{2.1}$$

$$\mu = \mu_l F + (1 - F)\,\mu_g \tag{2.2}$$

where the subscripts l and g respectively indicate liquid and gas phases.

2-1 Interface tracking algorithm

The time evolution of the interface topology is deduced from the advection equation given by:

$$\frac{\partial F}{\partial t} + \vec{\nabla}\left(\vec{V} F\right) = 0 \tag{2.3}$$

in which F represents the colour function and $\vec{V}$ the local flow velocity. We developped a new advection algorithm which allows to integrate F in time. The resolution is based upon geometrical hypothesis which ensures the interface's integrity and flux mass conservation. New straight lines are reconstructed in staggered grid corresponding to each face of the main cell. Within each staggered cell, an advected surface is determined which correspond to mass flux across the main cell face. The advection equation is implemented as shown on Fig.2,:

Figure 1: Quiescent water column

Figure 2: Advection algorithm

$$F^{m+1} = F^m - \Delta t \left(\frac{A_e - A_w + A_n - A_s}{\Delta x \Delta y} \right) + \Delta t F^m \left(\frac{u_e - u_w}{\Delta x} + \frac{v_n - v_s}{\Delta y} \right) \tag{2.4}$$

The subscripts m,n,w,s and e represent respectively the time step, and the faces of each cell designated by its cardinal points from the cell center 'north', 'west', 'south' and 'east'. The velocities of the main cell are noted on each face by the same subscripts. This algorithm allows to preserve mass conservation, and the new interface defined from staggered grid leads to a more accurate calculation of the new advected values of F^{m+1}.

2-2 Mathematical model

The motion of the two bubbles are described by the Navier-Stokes equation which is written in a non dimensionnal form as :

$$\vec{\nabla}.\vec{V} = 0 \tag{2.5}$$

$$\frac{\partial \rho \vec{V}}{\partial t} + \vec{\nabla} \left(\rho \vec{V} \otimes \vec{V} \right) = -\vec{\nabla} P + \frac{1}{\Re e} \vec{\nabla} \left[\mu \left(\vec{\nabla} \vec{V} + \vec{\nabla} \vec{V}^T \right) \right] + \rho \vec{g} + \frac{1}{Bo} \vec{F_{sv}} \tag{2.6}$$

with scales:

$$u^* = \frac{u}{U}, v^* = \frac{v}{U}, \text{with } U = \sqrt{gR_0}, \tag{2.7}$$

$$\rho^* = \frac{\rho}{\rho_g}, \mu^* = \frac{\mu}{\mu_g}, P^* = \frac{P - P_0}{\rho_l U^2}, \tag{2.8}$$

$$r^* = \frac{r}{R_0}, y^* = \frac{y}{R_0}, t^* = \frac{tU}{R_0}, \tag{2.9}$$

$$\Re e = \frac{\rho_l g^{1/2} R_0^{3/2}}{\mu_l}, Bo = \frac{\rho_l g R_0^2}{\sigma} \tag{2.10}$$

in which R_0 represents the radius of drop or bubble, $\otimes$ denote the inner product tensors, ρ and μ the fluid properties, P the pressure and $\vec{g}$ the gravity field. $\Re e$ and Bo are respectively Reynolds and Bond numbers. For better convenience, the * is omitted in Eqs.2.6. These equations are then discretised with MAC method on a fixed grid.

2-3 Surface tension contribution

The last term of equation (2.6) is the surface tension force $\vec{F_{sv}}$ which is modelled by the continuum surface tension force (CSF) developed by Brackbill et al., 1992. The surface force is transformed into volumic force by using a delta Dirac function centered on the interface. The surface tension force is expressed by the following relation:

$$\vec{F}_{sv} = \sigma \kappa \delta_s \vec{n} \tag{2.11}$$

in which σ represents the surface tension, δ_s the delta Dirac function, $\vec{n}$ the normal vectors to the interface and κ the curvature:

$$\kappa = -\vec{\nabla}_s.\vec{n} \tag{2.12}$$

This curvature is the surface divergence of the normal to an interface surface element. The normal vectors is determined by the non dimensionnal gradient of the colour function F. A smoothed colour function by convolution with a $\beta - spline$ kernel is commonly used to increase the accuracy of the normal and the numerical calculus of the curvature. We elaborated a new numerical curvature calculation method to improve numerical calculation of the surface tension's contribution, without using a smoothed colour function by CSF. This method based on the definition of two dimensionnal surface divergence and the calculation of the normal vector on each face and at a corner of a cell gives the curvature and superficial tension force values more accurately.

$$\kappa = \left| \frac{\partial \vec{n}}{\partial S} \right| \tag{2.13}$$

in which S is the length of the interface. The relation 2.13 is numerically approximated by the following expression :

$$\kappa = sign(\vec{n}_a \times \vec{n}_b)\sqrt{\left(\frac{n_{xb} - n_{xa}}{\Delta S}\right)^2 + \left(\frac{n_{yb} - n_{ya}}{\Delta S}\right)^2} \tag{2.14}$$

in which n_{xa}, n_{xb}, n_{ya} and n_{yb} are the normal vector when the positions are the nearest of thoses coordinates a and b, defined on the outline of the cell (Fig.3).

Figure 3: Curvature calculation

2-4 Projection Algorithm

The projection algorithm based on a discretization on MAC cell (Fig.4) allows to solve accurately the Navier-Stokes equations. The first step of this algorithm is the calculation of an explicit velocity. The final velocities are determined from the solenoidal nature of velocities by solving a Poisson equation of pressure correction, according to the following equations:

$$\vec{Q} = \left[-\vec{\nabla}\left(\rho\vec{V} \otimes \vec{V}\right) - \vec{\nabla}P + \frac{1}{\Re e}\vec{\nabla}\left[\mu\left(\vec{\nabla}\vec{V} + \vec{\nabla}\vec{V}^T\right)\right] + \rho\vec{g} + \frac{1}{Bo}\vec{F_{sv}}\right]^m \tag{2.15}$$

the new velocity $\tilde{\vec{V}}$ is defined by:

$$\tilde{\vec{V}} = \vec{V} + \frac{\Delta t}{\rho}\vec{Q} \tag{2.16}$$

and the Poisson equation for pressure correction $\tilde{P}$ is given by:

$$\vec{\nabla}\left(\frac{1}{\rho}\vec{\nabla}\tilde{P}\right) = \vec{\nabla}.\tilde{\vec{V}} \tag{2.17}$$

with

$$\tilde{P} = P^{m+1} - P^m \tag{2.18}$$

the pressure correction system is solved by the conjugate gradient method with an incomplete Cholevsky preconditionning (Meier, 1999, Rider et al., 1997).

Figure 4: Control volume or MAC cell

3 EXPERIMENTAL STUDY

A parrallelepipedic glass column with a square section of 15 cm sides and of 120 cm height is filled with water to 100 cm height (Fig.5). The water temperature is not controlled. So, it depends on the transfers with the ambient medium. Air is injected with a weak pressure ($\approx 1.2 \times 10^5 Pa$) through a PVC tube with a diameter section of 2 mm which is immerged in the water with an air injector situated at the bottom of the column. Two air-valves placed respectively here and there the injector allow to reduce or increase air pressure inside the immerged part of the tube as convenience. When the pressure is superior to hydrodynamic pressure, bubbles appear in the quiescent water. The system allows to control with relative accuracy the bubbles' production.

Bubbles and bubbles' coalescence are recorded with a camera Canon EOS D60 with an exposure time equal to 1/4000 s. The ISO speed rating is 800, The F-number is 3.5 and the focal length is 100 mm. The initial size of the picture is 3072x2048 pixels which allows to see on photos, details less than millimeter. The depth of field of our camera is less than 100 μm, so we have a great quality of photos.

The production of bubbles is continuous and the coalescences are induced by the rise of different bubble. The size of the bubbles is measured with a reference which is represented by the injector thickness (Fig.6).

4 RESULTS AND DISCUSSION

Several photos were registered during four hours, each of them allows to describe a type of coalescence (Fig.7). Photos reported on figures 6 and 7 a) represent strange bubbles' coalescence in which a liquid jet appears within the bubble. We consider that before the coalescence, the following bubble velocity increases because of the low drag forces induced by the wake effect of the leading bubble. However the immediatly neighbourring liquid on the bottom of the bubble still have inertia and its velocity is more important than those of the gas. Consequently the bottom interface of the finally bubble produced by the coalescence is pushed forward. For high velocity, this may cause a liquid jet within the bubble.

Different types of coalescence are presented on figures 7 b), c) and d). Our experimental device cannot control the reproductibility of the coalescence shape and several parameters as temperature, flow velocities, composition of water or injector outlet pressure must induce different coalescences. Each image represents various local parameters and it is very difficult to caracterize. However, most of these images (Fig.8) seem to show a mechanic interaction between both bubbles before coalescence. Each image, shows that there is an interface attraction during somme milliseconds before coalescence phenomenon (Fig.8). The coalescence of two bubbles carried out very quickly. So, we cannot deduce any pertinent comments of these observations. However, simulations carried out

Figure 5: Experimental system

with our numerical code lead to some coalescence in agreement with our experimental data (Fig.9). Simulations were executed with three bubbles regards of recorded photos, for different Reynolds number ranging from 10 to 2000, and for several Bond number. The ratio between air and water density and viscosity was respectively equal to 892 and 34. These values correspond to a water temperature equal to $40°C$. The water temperature is superior to the ambient because of the light source used to light the water column which is necessary to increase the photos quality. The water and air densities values used in our simulations are respectively equal to 992.22 and 1.112 $kg.m^{-3}$, the water and air viscosities are 653.25×10^{-6} and $19.2 \times 10{-}6$ $kg.m^{-1}.s^{-1}$ and the surface tension value is $\sigma = 0.06973$ $N.m^{-1}$. The radius of bubbles are initialized to $R_0 = 0.00275$ m for each bubble regards to figure 6. Each result of the simulation shows the presence of interface attraction between two bubbles at the same instant evaluated to $t = 70\Delta t$. The Reynolds and Bond numbers variation induce a modification of the interface topology during and after bubble coalescence and may produce different type of bubbles' coalescence comportment.

Our numerical code is able to simulate the rise of two air-bubbles by buoyancy forces and their coalescence led by the wake effect. However, some of our results are not in agreement with those of the photos, because experimental conditions are not well defined. We noticed that the discretization of convective terms of the Navier Stokes equations induces different type of coalescence bubble and interface topology. For instance, for an initial distance between two bubbles D=0.33R_0 (Fig.11), we used convective terms discretization based on MAC cells (Fig.4). Therefore, any derivative of a flux Φ may be written as

$$\left.\frac{\partial \phi}{\partial x}\right|_{x_{i,j}} = \frac{\Phi_{i+\frac{1}{2},j} - \Phi_{i-\frac{1}{2},j}}{\Delta x} \tag{4.1}$$

This may be generalized as :

$$\left.\frac{\partial \phi}{\partial x}\right|_{x_{i,j}} = \frac{\left[\alpha\Phi_{i+\frac{1}{2},j+1} + \beta\Phi_{i+\frac{1}{2},j} + \alpha\Phi_{i+\frac{1}{2},j-1}\right] - \left[\alpha\Phi_{i+\frac{1}{2},j+1} + \beta\Phi_{i+\frac{1}{2},j} + \alpha\Phi_{i+\frac{1}{2},j-1}\right]}{(2\alpha+\beta)\,\Delta x} \tag{4.2}$$

In which α and β represent the contribution of neighbouring cells. So, there is a different discretization for each values α and β. The discretization reported in (4.1) is obtained for a couple (α,β)=(0,1). For a couple (α,β)=(1,2), we used Young & Parker method for the calculus of normal vectors which is the gradient of the colour function. The variation of α, β values corresponds to different comportment of the bubbles coalescence. So these values

Figure 6: Coalescence between two air-bubbles rising in a quiescent water

induce different shape at the end of the coalescence. The figure 12 presents the bubble shape after the coalescence for different couple (α,β) and a non-dimensionnal time step equal to 0.0002. Numerical results are considered at the same instant, t=600Δt. The images show that results are depending of the different type of discretization of the convective terms of the Navier-Stokes equations. After the coalescence, experimental images show that the water penetrates by the bottom of the bubble. Simulations lead to the same phenomenon with the use of an adapted convective discretization regards to figure 12 and 10.

5 COMMENTARY

Figures 12 shows the effect of the type of discretization on the shape of bubble coalescence. Consequently the use of MAC sheme must be improved to take account of the impact of the discretization of the convective terms. We observe that any type of numerical code, at our knowledge, may simulate correctly air bubble coalescence, because of various unknown initial parameters and unknown phenomena. In the same way, it is very difficult to produce the same experimental conditions of air bubbles' coalescence in water. Our numerical study is based on the hypothesis of a quiescent water. This condition seems to be impossible to realize experimentaly. Furthermore, it is difficult to caracterize numericaly and experimentaly the beginning of the bubbles' coalescence. In fact, the coalescence phenomenon induce, the knowledge of various initial parameters which are very difficult to define experimentaly. So we cannot conclude that our numerical code leads to a better accuracy in the case of bubbles' coalescence. Moreover, we do not applied our numerical code for different initial bubbles shapes. In the study of bubble's coalescence, the initial shape is usualy described with a circular form. For instance, the figure 6 shows that any initialisation may produce intense liquid jet in the bubble. This may be due to an incomplete mathematical formulation or chaotic experimental condition.

6 CONCLUSION

Numerical and experimental studies of air-bubbles coalescence in a quiescent water has been carried out with an accurate and robust numerical method. Experimental and numerical results are in good agreement. The numerical study concerns the interface attraction before bubble coalescence. Therefore, the effect of the convective terms discretization of Navier-Stokes equations on the comportment of bubble coalescence have been investigated. Results show that the liquid jet intensity under the bubble produced after coalescence depends on the type of discretization of the convective terms. Any intense liquid jet during the coalescence may be produced numerically.

Figure 7: Experimental visualisation of coalescence between two air-bubbles

References

[1] Brackbill J.U., Kothe D., Zemach C., " A continuum method for modeling surface tension", Journal of computational physics, Vol. 100, pp 335-354, (1992).

[2] Chen L., Garimella S.V., Reizes J.A., Leonardie., " Analysis of bubbles rise using the VOF method : I. isolated bubbles", HTD-Vol. 326, National Heat Transfer Conference, Volume 4, ASME (1996).

[3] Chen L., Garimella S.V., Reizes J.A., Leonardie., " Analysis of bubbles rise using the VOF method : II. bubbles interactions, wall effects and evaporation", HTD-Vol. 326, National Heat Transfer Conference, Volume 4, ASME (1996).

[4] Chen L., Li Y., " A numerical method for two-phase flows with an interface", Environmental Modelling & Sofware 13, pp 247-255, (1998).

[5] Chen L., Li Y., Manasseh R., " The coalescence of bubbles - A numerical study", Third International Conference on Multiphase Flow, ICMF'98, Lyon , France, June 8-12, (1998).

Figure 8: Attraction phenomenon before coalescence

[6] Hirt C.W., Nichols B.D., " Volume of fluid (VOF) Method for the dynamics of free boundaries", Los Alamos Scientific Laboratory, Los Alamos, New Mexico, (1979).

[7] Li J., " Calcul d'interface affine par morceaux (Piecewise Linear Interface Construction)", C.R. Acad.Sci. II B 320, pp 391-396, (1995).

[8] Meier M., " Numerical and experimental study of large steam-air bubble injected in a water pool", Thesis of Technical Sciences from the Swiss Federal Institute of Technology Zurich, (1999).

[9] Meier M., Yadigaroglu G., Smith B.L., " A novel technique for including surface tension in PLIC-VOF methods", Eur.J.Mech.B/Fluids 21, pp. 61-73, (2002).

[10] Narayanan S., Goossens L.H.J., Kossen N.W.F., " Coalescence of two bubbles rising in line at low Reynolds number", Chemical Engineering Science, Vol.29, pp. 2071-2082, (1974).

[11] Noh W. and Woodward P., " SLIC (Simple Line Interface Calculation)", In van de Vooren A. and Zandbergen P., editors, Proceedings, Fifth International Conference on Fluid Dynamics, Volume 59 of Lecture Notes in Physics, pp 330-340, Berlin Springer, (1976).

[12] Rider J.W., Kothe D.B., Mosso S.J., Cerutti J.H., " Accurate solution algorithms for incompressible multiphase flows", Los Alamos National Laboratory, Los Alamos, New Mexico, 87545, USA and John I.

Figure 9: Attraction phenomenon before coalescence Δt=0.5 ms, $\Re e$=705, Bo=1, $\frac{\rho_l}{\rho_g}$=892, $\frac{\mu_l}{\mu_g}$=34

Hochstein Mechanical Engineering Department, Memphis State University, Memphis, TN, 38152, USA, (1995).

[13] Rider J.W., Kothe D.B., " Reconstructing Volume tracking", Los Alamos National Laboratory, Los Alamos, New Mexico, 87545, USA, (1997).

[14] Rider J.W., Kothe D.B., Mosso S.J., Brock J.S., " Volume tracking of interface having surface tension in two and three dimensions", Los Alamos National Laboratory, Los Alamos, New Mexico, 87545, USA, (1995).

[15] Tse K.L., Martin T., Mc Farlane C.M, Nienow A.W.," Small bubble formation via a coalescence dependent break-up mechanism", Chemical Engineering science 58, pp. 275-286, (2003).

[16] Zaleski S., " Computation of multiphase flow by volume of fluid and high-order front tracking methods", Rapport d'activité de modélisation en mécanique, Université Pierre et Marie Curie, Paris VI, (1999).

Figure 10: Shape after coalescence

Figure 11: Distance initialization $\frac{D}{R_0}$=0.33, R_0=0.00275 m

Figure 12: Dependance of discretization couple after coalescence