

An Eulerian-Lagrangian open source solver for bubbly flow in vertical pipes

C. Peña-Monferrer^a, J. L. Muñoz-Cobo^a, G. Monrós-Andreu^b, R. Martínez-Cuenca^b, S. Chiva^{b,*}

^a*Institute for Energy Engineering, Universitat Politècnica de València. Camí de Vera, s/n, 46022 València, Spain*

^b*Department of Mechanical Engineering and Construction, Universitat Jaume I. Campus del Riu Sec, 12080 Castelló de la Plana, Spain*

Abstract

Air-water two-phase flow is present in natural and industrial processes of different nature as nuclear reactors. An accurate local prediction of the boiling flow could support safety and operation analyses of nuclear reactors. A new Eulerian-Lagrangian approach is investigated in this contribution. A new solver has been developed and implemented in the framework of the open source package OpenFOAM[®] and based on the PIMPLE algorithm coupled with the Lagrangian equation of motion has been implemented for computing incompressible bubbly flows. Each bubble is divided in equivolumetric volumes and tracked into the Eulerian mesh for an appropriate assignment of the effect of the bubble in the cell without resolving the interface. The coupling between phases is done considering in the momentum equation the interfacial forces and bubble induced contribution along the bubble path during an Eulerian time step. The bouncing of the bubbles between themselves and the wall is modeled with a dynamic soft sphere model. The computational results obtained for different flow conditions are validated with the recently released experimental data on upward pipe flow. The test section used is a 52 mm pipe of 5500 mm of length maintained under adiabatic conditions with air and water circulating fluids working with inlet velocity ranges of 0-2 m/s and 0-0.3 m/s for the continuous and dispersed phase respectively. Averaged results of radial distribution for void fraction, chord length, turbulence kinetic energy, dispersed and continuous velocity profiles show a good agreement among different flow conditions.

Keywords: Lagrangian-Eulerian, CFD, OpenFOAM, Two-phase flow, Bubbly flow, Unstructured grid, Soft-sphere collisions

1. Introduction

Due to its importance and challenges related to predict the flow structure using computational methods, two-phase flow have been investigated over the years being a research focus with significant strides accomplished to date. Computational simulations employing very different approaches

*Corresponding author

Email addresses: cmonfer@upv.es (C. Peña-Monferrer), jlcobos@iqn.upv.es (J. L. Muñoz-Cobo), gmonros@emc.uji.es (G. Monrós-Andreu), rcuenca@emc.uji.es (R. Martínez-Cuenca), schiva@emc.uji.es (S. Chiva)

were used to predict the two-phase flow behavior at different levels [1, 2]. The choice of a specific method depends on its suitability for the range of operating conditions, the scale and the complexity of the domain to investigate. The literature encompasses developments at microscopic, mesoscopic and macroscopic levels as Lattice Boltzmann, discrete particle or two-fluid models respectively.

This research, focused at the mesoscopic level, developing a discrete particle method using an Eulerian-Lagrangian (EL) approximation with the aim of investigate in detail the effect of two phase flow in nuclear power plants as e.g. fuel assemblies. The EL approach started some decades ago for the mathematical simulation of sprays, O'Rourke [3, 4] developed a new approach coupling the Lagrangian equation for droplet distribution function of [5] with an Eulerian description, in their calculations velocity and pressure were obtained by means of the Navier-Stokes equations while the motion of each particle were solved using the Newton's second law as well as the particle-particle, particle-wall, coalescence and breakup during the particle path. Lately Dukowicz [6] developed a EL two-way coupling including momentum coupling and volume effects. For the most part, this method have been applied to sprays or particle-laden flows. Further and in a lesser extent EL were applied to bubbly flows applications [7] and [8, 9]. The reader is referred to Subramaniam [10] for details about LE methods.

Although in EL methods the interface between phases is not resolved and the sphericity of the bubble is assumed, this approach plays a useful role modelling particle-particle interactions, particle-wall interactions, coalescence and breakup with the possibility to compute a large number of bubbles without the need to resolve the interface between the phases. On the other hand, the fact of computing individual bubbles allows the use of interfacial force coefficients usually derived for individual spheres while for two-fluid approaches its applicability is questionable.

Experimental work has been previously done to analyze the two-phase flow behavior and to assess the validity of existing and novel computational methods as in Muñoz-Cobo et al. [11] and Chiva et al. [12]. In this contribution the computational results obtained for different flow conditions are validated by comparison with recently released experimental data concerning an air-water system [13], in an upward vertical pipe, including the transition from bubbly to slug flow scenarios. The scenario and techniques used to obtain the experimental data fulfills the following requirements and in the view of the authors is appropriate to validate the approach investigated:

- The domain is known with a well-known boundary conditions and its validity is extensible to a large number of industrial applications.
- The tests include different flow patterns and transition regions.
- The data has enough information to test the involved models with high spatial resolution at different axial locations for the dispersed field as velocity, void fraction, chord lengths and number of bubbles and for the carrier phase as velocity and turbulence.

With such considerations this solver could be tested afterwards in large industrial domains involving two-phase flow as e.g.: nuclear fuel assemblies, chemical reactors, stirred tanks, biological reactors in wastewater treatment plants. Nevertheless, for transitional regimes becoming separated flows particular attention will be paid in future contributions.

paper.

2. Governing equations

The motion of each single bubble is computed by solving the equations of motion in a Lagrangian frame:

$$m_b \frac{dU_b}{dt} = \sum_{b=1}^n F_b \quad (1)$$

The total force F_b acting on a single bubble is given by the sum of various forces modeled as a function of flow parameters:

$$F_b = F_{\text{gravity}} + F_{\text{virtual mass}} + F_{\text{drag}} + F_{\text{lift}} + F_{\text{collisions}}, \quad (2)$$

with the drag modeled as [Ishii and Mishima \[14\]](#), virtual mass as [Drew and Lahey \[15\]](#), and lift [Auton et al. \[16\]](#). The collisions producing the bubble-bubble and bubble-wall bouncing are modeled as a spring mass. The model is based on the existing soft sphere models [17] [18] with a new model in agreement with the elastic behavior of the bubbles in the deformation behaving as a Hookean spring as described in the Fig. 1. In order to calculate the stiffness of the spring in a single bubble (b_i) with radius R_i , a_i and b_i the semi-minor axis and semi-major axis respectively, we will determine the stiffness in each bubble as a function of the increase of the surface area due to the bubble deformation. Assuming that the deformation of the bubble conserves the volume, the surface energy increase due to surface deformation of the spherical bubble into an oblate spheroid: $\Delta E = \sigma(S_{\text{oblate}b_i} - S_{\text{sphere}b_i}) = \frac{1}{2}K_{b_i}(R_{b_i} - a_{b_i})^2$. The value of K_{b_i} computed in a similar way as in [19], based on the expression for an oblate surface.

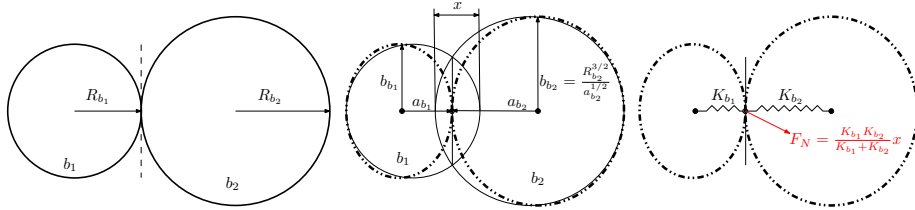


Figure 1: Spring scheme in bubble deformation

The instantaneous liquid velocity (u_l) used to compute the forces is computed as the sum of the mean liquid velocity U_1 and the fluctuating velocity component $u'_l(t)$. As previously done by several researchers as in [20], $u'_l(t)$ is predicted with a discrete random walk (DRW) model [21]. The mean liquid velocity is solved with the Navier-Stokes liquid-phase momentum conservation equation coupled to the Lagrangian to include the liquid volume fraction ($\alpha_l = 1 - \alpha_g$) and the momentum source contribution, M_d , generated by each bubbles along its path on the cell. Obtained from the motion equation, it represents the force from the disperse phase on the fluid phase per unit mass of fluid.

$$\frac{\partial}{\partial t} \alpha_l U_1 + \nabla \cdot (\alpha_l U_1 U_1) = -\alpha \nabla p_{rgh} - \alpha (g \cdot x) + \nabla \cdot \tau + M_d \quad (3)$$

A standard κ - ϵ turbulence model [22] is used to model the effect of turbulent fluctuations in the carrier phase.

Finally, the bubbles rising in the pipe expand because of the pressure changes. This bubble size variation is calculated based on the gas and liquid pressure differences and the ideal gas law, being evaluated each movement and updating the bubble diameter.

3. Code implementation

The proposed solver is based on the PIMPLE algorithm established in the released versions coupled with a generic Lagrangian tracking integrated with the Verlet Leapfrog algorithm. Deep modifications have been carried out to compute bubbly flow.

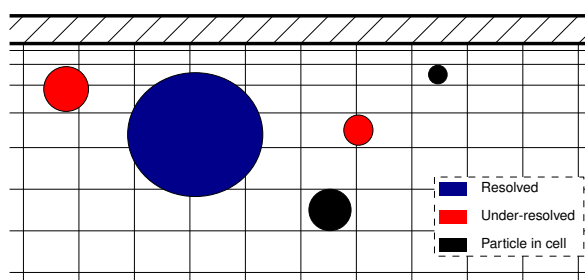


Figure 2: Different bubble sizes on a fixed Eulerian grid

A variation of the algorithm shown in [23] [24] is implemented in OpenFOAM to track the bubbles along the cells. The motion of bubbles is done for in unstructured, arbitrary polyhedral meshes leading with 3D meshes of complex geometries including the tracking in a parallel decomposed case. In this context also parallel evaluation of collision forces has been included [25].

The problem of effectively distribute the source term M_d and void fraction in the cells for resolved bubbles is usually addressed with template functions as in [26]. However for arbitrary unstructured grids this assignment can give unrealistic or non-accurate results. Instead, in this contribution the sphere is divided in several fictitious points inside each bubble and tracked with the center of mass. The distribution of this points is done strategically with a equivolumetric partitioning of the sphere with the algorithm shown in [27]. Each bubble k has N_s subpoints, the contribution of all of the subpoints, N_{inCell} , which have been in a cell during a residence time dt .

The models for the expansion of the bubble, a new bubble injection algorithm and the soft sphere collision haven been implemented.

4. Description of the experimental facility

The experimental facility is located at Universitat Jaume I, Spain. Experiments has been carried out by using an upward flow experimental loop with inner diameter of 52 mm and 5500 m length. Operating fluids were purified water ($\sim 30 \mu S/m$) and air, mixed at bottom of the section in a mixing chamber through 4 sparger (mean porosity $\sim 40 \mu m$). The measurement system consisted of three mounted four-sensor conductivity probes, mechanical traversers, a measurement

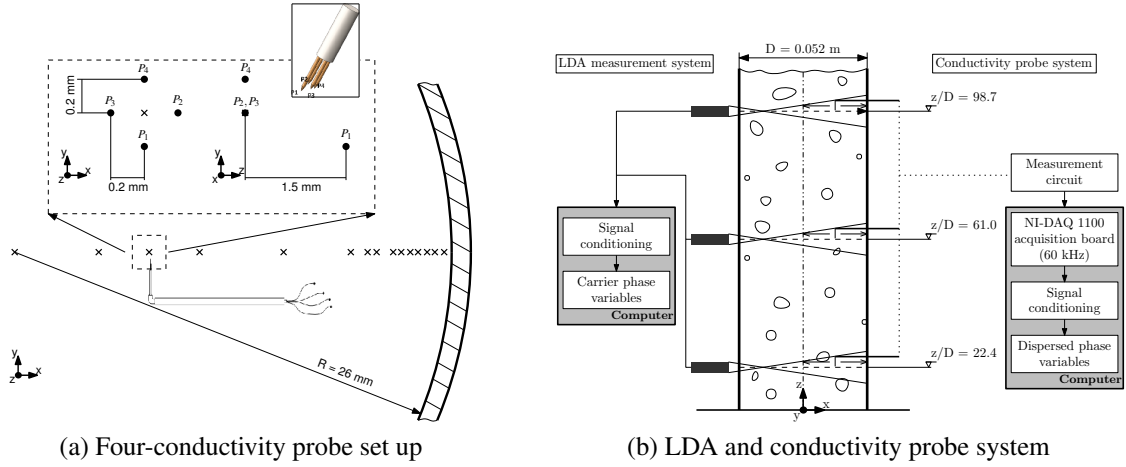


Figure 3: Experimental configuration to obtain the two-phase flow variables

circuit, a digital high-speed acquisition board, and the software used to signal processing. The four-sensor conductivity probe was attached to the mechanical traverser mounted on a specially designed flange, and it could be moved along the radial direction of the test section using controlled step motors. The measurement circuit was used to measure the potential difference between the exposed tip and the grounded terminal. A high-speed NI SCXI-1000 acquisition board and a PC were used to acquire the voltage signal of the four-sensor probe, with the help of a control program developed under NI LabView software environment.

5. Virtual four-conductivity probes system for dispersed phase data calculation

To obtain the flow characteristics of the dispersed phase is not trivial. The well-established conductivity probes system used in the experiments has been adapted to the CFD code. This represents a new methodology from the point of view of the simulations.

Conductivity probes basically act as a phase identifier. Applied to the experiments, the probe is connected to a power supply with a fixed voltage, due to the large difference in conductivity between the liquid phase and the gas phase, the impedance signal acquire vary depending on phase surrounding tips. When the tip is surrounded by liquid, a lower voltage is put out; and when the tip contacts with gas, a higher voltage is obtained. But due to the finite size of each sensor and the time delay needed to wet or rewet the sensor tips, the output signal (Fig. 4a) of the four-sensor probe differs from ideal two-state square-wave and the signal is processed to obtain the regenerated square-wave signals as shown in Fig. 4b. In the simulations the square-wave (binary) signal is obtained directly. Afterwards, this signal will be processed with the same code as for the experiments to obtain the void fraction, velocity, chord length and number of bubbles detected.

6. CFD setup

In order to validate the solver, a pipe of 1.955 m of length corresponding to the distance between two measurement sections ($z/D = 61$ and $z/D = 99$) is modeled and simulated. The mesh

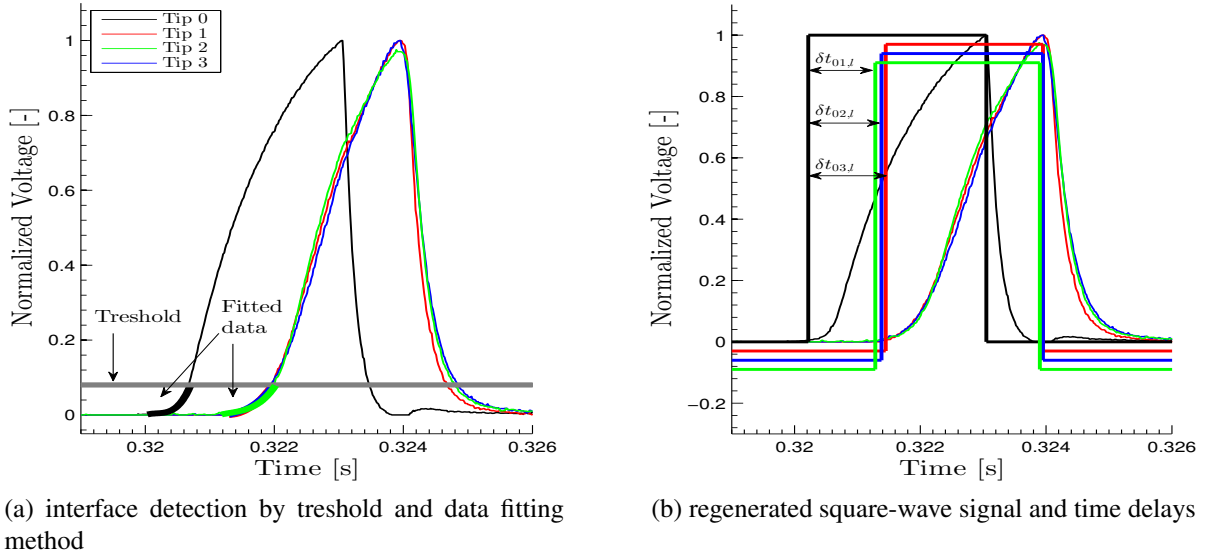


Figure 4: Signal processing

was created with the native OpenFOAM[®] mesh generation tool *blockMesh*. After the mesh sensitivity analysis was carried out, a mesh of 938,400 elements with 5 mm of axial distance between nodes and 35 radial nodes. Water and air properties are calculated as a function of the operating temperature assuming tap water.

7. Results

The predictive capability of the solver is assessed by comparing the flow characteristics with radial profiles at $z/D = 99$ for a scenario with liquid velocity of 1 m/s, superficial gas velocity of 0.05 m/s and 5% of void fraction. The technique described in Section 5 allows us to obtain additional computational results as dispersed phase velocity, chord length or bubbles detected. In Fig. 5 the first results of this solver are shown with an overall good agreement with the experimental results. This results are promising, further work will be done to validate the solver under a wide range of flow conditions.

8. Conclusions

An Eulerian-Lagrangian solver to simulate polydisperse bubbly flows in adiabatic vertical pipes was discussed and successfully implemented into OpenFOAM[®].

Satisfactory agreement between experimental measurements and numerical results was obtained for the void fraction profiles, chord lengths, dispersed phase velocity and number of bubbles detected. Future work will be focused on testing the solver validity on regime transitions and in flow conditions with higher inlet dispersed phase void fraction. In addition, subcooled boiling with EL techniques will be investigated to analyze the two phase flow effect in fuel assemblies.

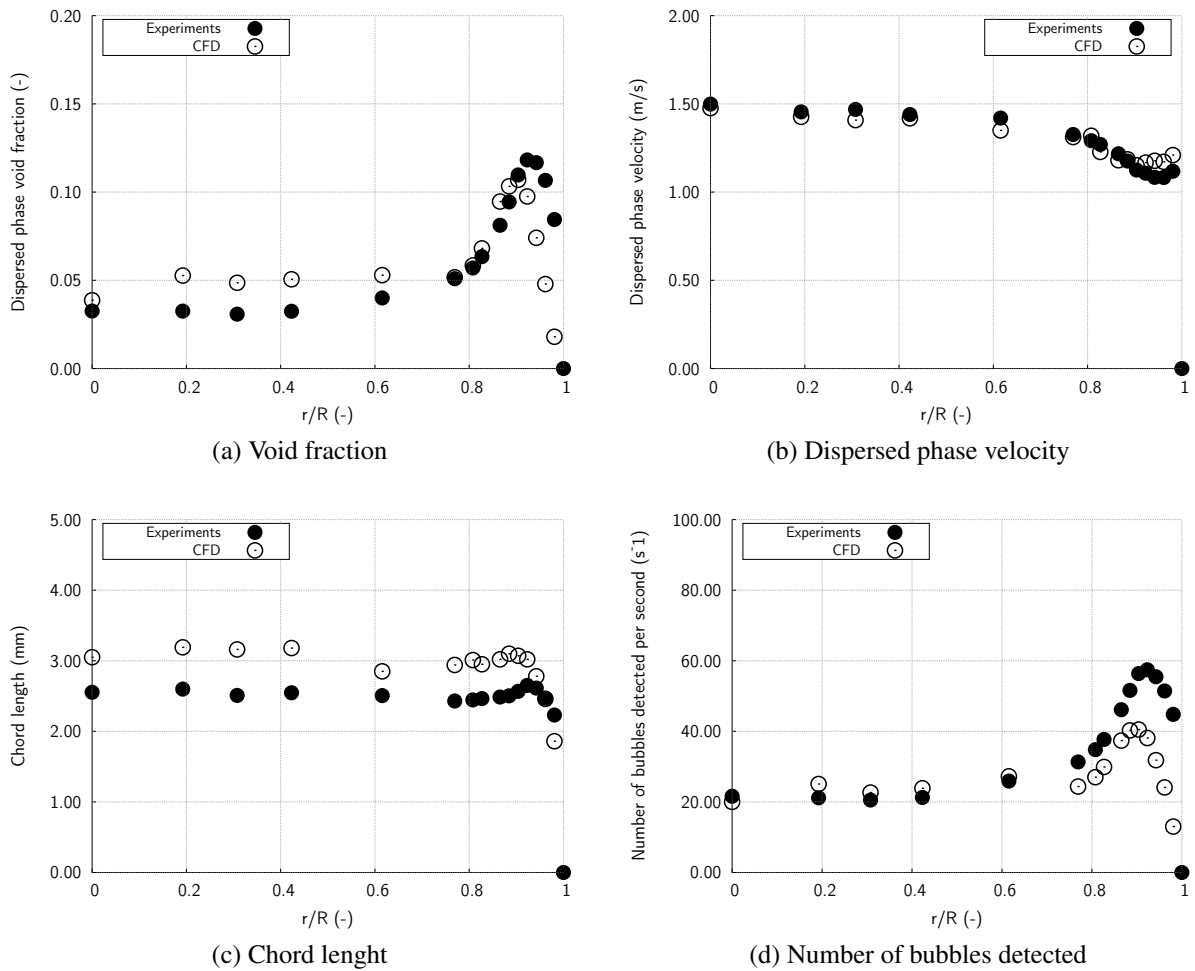


Figure 5: CFD comparison with experimental results

Acknowledgements

The authors sincerely thank the “Plan Nacional de I+D+i” Project EXPERTISER ENE2010-21368-C02-01 and ENE2010-21368-C02-02 for funding the project.

References

- [1] M. Ishii, T. Hibiki, Thermo-fluid Dynamics of Two-Phase Flow, Smart Energy Systems, Springer, 2006.
- [2] A. Prosperetti, G. Tryggvason, Computational Methods for Multiphase Flow, Cambridge University Press, 2007.
- [3] P. O’Rourke, Collective drop effects on vaporizing liquid sprays, Ph.D. thesis, 1981.
- [4] P. O’Rourke, The kiva computer program for multidimensional chemically reactive fluid flows with fuel sprays, in: R. Glowinski, B. Larroutou, R. Temam (Eds.), Numerical Simulation of Combustion Phenomena, volume 241 of *Lecture Notes in Physics*, Springer Berlin Heidelberg, 1985, pp. 74–89.
- [5] F. A. Williams, Spray combustion and atomization, *Physics of Fluids* (1958-1988) 1 (1958) 541–545.
- [6] J. K. Dukowicz, A particle-fluid numerical model for liquid sprays, *Journal of Computational Physics* 35 (1980) 229–253.

- [7] D. Z. Zhang, A. Prosperetti, Ensemble phase-averaged equations for bubbly flows, *Physics of Fluids* 6 (1994) 2956–2970.
- [8] E. Shams, J. Finn, S. Apte, A Numerical Scheme for Euler-Lagrange Simulation of Bubbly Flows in Complex Systems, *International Journal for Numerical Methods in Fluids* (2010).
- [9] M. A. A. E. A. Essa, Coupled Lagrange-Euler model for simulation of bubbly flow in vertical pipes considering turbulent 3D random walks models and bubbles interaction effects, Ph.D. thesis, 2012.
- [10] S. Subramaniam, Lagrangian–Eulerian methods for multiphase flows, *Progress in Energy and Combustion Science* 39 (2013) 215–245.
- [11] J. L. Muñoz-Cobo, S. Chiva, M. Essa, S. Mendes, Experiments performed with bubbly flow in vertical pipes at different flow conditions covering the transition region: simulation by coupling eulerian, lagrangian and 3d random walks models, *Arch. Thermodyn.* 33 (2012) 3–39.
- [12] S. Chiva, J. E. Julia, L. Hernandez, S. Mendez, J. L. Muoz-Cobo, A. Romero, Experimental study on two-phase flow characteristics using conductivity probes and laser doppler anemometry in a vertical pipe, *Chem. Eng. Commun.* 197 (2009) 180–191.
- [13] G. Monrós-Andreu, S. Chiva, R. Martínez-Cuenca, S. Torró, J. E. Juliá, L. Hernández, R. Mondragón, Water temperature effect on upward air-water flow in a vertical pipe: Local measurements database using four-sensor conductivity probes and LDA, *EPJ Web Conf.* 45 (2013) 01105.
- [14] M. Ishii, K. Mishima, Two-fluid model and hydrodynamic constitutive relations, *Nucl. Eng. Des.* 82 (1984) 107–126.
- [15] D. A. Drew, R. T. Lahey, The virtual mass and lift force on a sphere in rotating inviscid flow, *Int. J. Multiphase Flow* 13 (1987) 113–121.
- [16] T. R. Auton, J. C. R. Hunt, M. Prud’Homme, The force exerted on a body in inviscid unsteady non-uniform rotational flow, *J. Fluid Mech.* 197 (1988) 241–257.
- [17] Y. Tsuji, T. Kawaguchi, T. Tanaka, Discrete particle simulation of two-dimensional fluidized bed, *Powder Technology* 77 (1993) 79–87.
- [18] P. A. Cundall, O. D. L. Strack, A discrete numerical model for granular assemblies, *Géotechnique* 29 (1979) 47–65.
- [19] A. Sato, M. Shirota, T. Sanada, M. Watanabe, Modeling of bouncing of a single clean bubble on a free surface, *Physics of Fluids* 23 (2011) 013307.
- [20] V. V. Buwa, D. S. Deo, V. V. Ranade, Eulerianlagrangian simulations of unsteady gasliquid flows in bubble columns, *International Journal of Multiphase Flow* 32 (2006) 864 – 885.
- [21] A. Gosman, E. Loannides, Aspects of computer simulation of liquid-fueled combustors, *Journal of Energy* 7 (1983) 482–490.
- [22] B. Launder, D. Spalding, The numerical computation of turbulent flows, *Comput. Method. Appl. M.* 3 (1974) 269–289.
- [23] A. M. Vaidya, P. M. V. Subbarao, R. R. Gaur, A Novel and Efficient Method for Particle Locating and Advancing over Deforming, NonOrthogonal Mesh, *Numerical Heat Transfer, Part B: Fundamentals* 49 (2006) 67–88.
- [24] G. B. Macpherson, N. Nordin, H. G. Weller, Particle tracking in unstructured, arbitrary polyhedral meshes for use in CFD and molecular dynamics, *Communications in Numerical Methods in Engineering* 25 (EMPTY) 273.
- [25] G. Macpherson, J. Reese, Molecular Dynamics in Arbitrary Geometries: Parallel Evaluation of Pair Forces, *Molecular Simulation* 34 (2008) 97–115.
- [26] N. Deen, M. van Sint Annaland, J. Kuipers, Multi-scale modeling of dispersed gasliquid two-phase flow, *Chemical Engineering Science* 59 (2004) 1853 – 1861. *Complex Systems and Multi-scale Methodology*.
- [27] G. Yang, I.-M. Chen, Equivolometric partition of solid spheres with applications to orientation workspace analysis of robot manipulators., *IEEE Transactions on Robotics* 22 (2006) 869–879.