Computational simulation of Taylor bubbles motion in a stagnant liquid inside a vertical column

M. B. de Azevedo[1], F. R. T. do Nascimento[1], J. L. H. Faccini[1], M. L. Moreira[1] and J. Su[2]
E-mail: bertrand@ien.gov.br

[1] *SETER, IEN* [2]*PEN, COPPE, UFRJ*

Keywords: computational simulation; Taylor bubbles; Open FOAM; stagnant liquid column; vertical flow.

In the present work [1], the motion of single Taylor bubbles in a vertical column of stagnant liquid was computationally simulated by using a recent version of the open source software Open FOAM, where a newly developed method (isoAdvector) was introduced together with an improvement of the InterFOAM solver, named Interflow. The simulated column consists of a tube with 2 m in length and inner diameter of 0.024 m, sealed at the ends and partially filled with water. Experiments was also performed under these conditions (Figure 1) in order to measure some bubble parameters by using an ultrasonic technique [2]. The computational simulation was performed solving a mathematical model formed by the isothermal, incompressible and laminar Navier-Stokes equations [3]. In addition, it was assumed that the fluids are immiscible and no turbulence models were used. The equations were discretized by the VOF (volume of fluid) method and solved using the Gauss iteration method. Numerical solutions were obtained for the bubble velocities and shapes which were compared with the experimental results and correlations from literature.

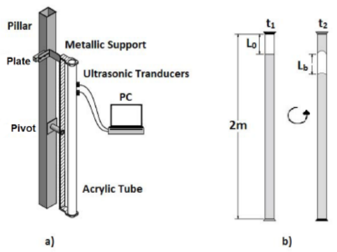

Figure 1. Schematic of the test section

Initially, simulations were performed using nonparametric meshes. The results indicated a discrepancy between the simulated and experimental results for the velocities and shapes of the bubbles. Despite these disagreements, the simulated bubbles took the typical Taylor bubble profile, which was not achieved in previous attempts to simulate this type of bubble using the InterFOAM solver.

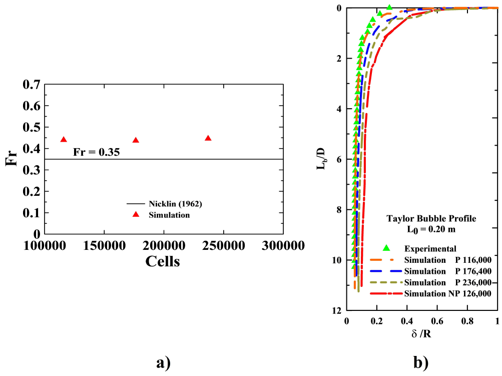

Figure 2. Experimental and simulated results using parametric mesh with different number of cells: a) bubble velocities; b) bubble profiles

New simulations were performed using parametric meshes with different number of cells. The simulated bubble velocities did not change with the increase of the number of parametric cells (Figure 2a). However, when the number of cells increased, the bubble profile tended to move away slightly from the experimental profile, especially near to the nose region (Figure 2b). Despite the simulated results did not present a good agreement with the experimental ones, the Interflow solver of OpenFOAM seems to be a promising tool for computational simulations of Taylor bubble motion. However, a better knowledge about this solver is needed to perform a more adequate mesh construction and propose modifications to improve the modeling of these bubbles motion, and consequently the simulated results concerning the bubble velocities and shapes.

References

[1] NASCIMENTO, F.R.T.; DE AZEVEDO, M. B.; FACCINI, J. L. H.; MOREIRA, M. L.; SU, J. Computational simulation of Taylor bubbles motion in stagnant liquid inside a vertical column. In: Proceedings of The Brazilian Congress Of Thermal Sciences And Engineering, Águas de Lindóia, SP, Brazil, 2018.

[2] DE AZEVEDO, M. B.; DOS SANTOS, D.; FACCINI, J. L. H.; SU, J., Experimental study of the falling film of liquid around a Taylor bubble. In: International Journal of Multiphase Flow, v. 88, p. 133-141, 2017.

]3] ROENBY, J.; BREDMOSE, H.; JASAK, H. A computational method for sharp interface advection. In: Royal Society Open Science, v. 11, n. 3, 2016.