Computational simulation of a single Taylor bubble rising in a vertical column with stagnant liquid

F. R. T. do Nascimento[1], M. B. de Azevedo[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*

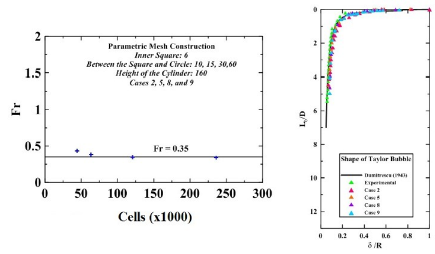
Keywords: Taylor bubbles; stagnant liquid; bubble velocity; bubble shape; Open FOAM.

Computational simulation of the motion of single Taylor bubbles in a vertical column with stagnant liquid was performed in the open source software Open FOAM using the InterFlow solver and the IsoAdvector method [1]. Numerical solutions were obtained for the rise velocities and shapes of the bubbles, which were compared with experimental data [2] and correlations from literature. The two fluids were assumed immiscible. The governing equations were discretized by the volume-of-fluid (VOF) method and solved using the Gauss iteration method. Parametric mesh was used in order to improve the modeling of curvilinear geometry.

Table 1. Schematic of the test section.

Case	Cells at the inner square	Cells between the square and the circle	Cells at the height of the cylinder
1	4	15	160
2	6	10	160
3	6	15	40
4	6	15	80
5	6	15	160
6	6	15	320
7	6	15	640
8	6	30	160
9	6	60	160

In order to create a parametric cylindrical mesh, it was necessary to define a number of parameters: the radius and the height of the cylinder, the number of cells at a inner square contained in its base, between this inner square and the circle (cylinder base) and in the cylinder height [3]. The cylinder radius and height were defined by the characteristics of the tube where the bubble movement was simulated. In the present work, parametric meshes were created varying the other parameters cited above, in order to obtain information about the influence of each of them on the simulated results (Table 1).



a) b)

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

The results indicated that the IsoAdvector method of the Interflow solver proved to be a powerful tool for computational simulation of the movement of Taylor bubbles and for the analysis of the hydrodynamic characteristics of slug flow in tubes. The method was shown to determine accurately the interface between the fluids, resulting in an excellent prediction of the velocity and shape of the bubbles. However, the mesh construction for a good performance simulation must be done with criteria. In the present work, the best results were obtained for mesh constructions with a minimum number of 6 cells at the inner square, 30 cells between the square and the circle and 160 cells in cylinder height. Naturally, a definition of the best mesh construction for a given simulation must take into account the balance between the accuracy of results and the computational costs.

References

[1] NASCIMENTO, F.R.T and DE AZEVEDO, M. B. and FACCINI, J. L. H. and MOREIRA, M. L. and SU, J., Computational simulation and experimental study of a single Taylor bubble rising in a vertical column with stagnant liquid. In: Proceedings of the International Nuclear Atlantic Conference, Santos, SP, Brazil, 2019.
[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] NASCIMENTO, F.R.T and DE AZEVEDO, M. B. and FACCINI, J. L. H. and MOREIRA, M. L. and 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.