Thermal characteristics of Taylor bubble flow

Hassan Shaban, Stavros Tavoularis

Department of Mechanical Engineering, University of Ottawa, Ottawa, Canada

Abstract

Results of numerical simulations of Taylor bubbles with turbulent wakes rising in stagnant water columns are presented. The velocity field was found to be in good agreement with previous measurements, collected using PIV. Heating of the tube wall with a constant heat flux permitted the study of the temperature field. It was found that the wall temperature and the convective heat transfer coefficient underwent large fluctuations as a Taylor bubble passed by.

Keywords: Taylor bubble, two-phase, slug flow, turbulence, IDDES

1. Introduction

Slug flow is a two-phase gas-liquid flow regime, in which the gas phase appears mostly in the form of relatively large *Taylor bubbles*, which have characteristic bullet-like shapes and occupy most of the channel cross-section. These large bubbles are separated by liquid slugs, which may contain much smaller quasi-spherical or ellipsoidal gas bubbles (Figure 1). In vertical upwards slug flow, the net liquid flow is always upwards, but liquid may be falling downwards in the annular sections that surround the Taylor bubbles. Slug flow occurs in pipes for wide ranges of flow rates and void fraction [1], but not when the pipe diameter exceeds a certain threshold, whose value depends on the fluid properties [2].

Slug flows are encountered in a variety of industrial systems, including oil and gas transport, nuclear safety systems and heat exchangers in power plants. In some applications, slug flow effects can be detrimental, as they introduce flow pulsations, which may cause pressure or thermal cycle damage. In other cases, the slug flow regime would be desirable; for example, in petroleum extraction using the gas-lift method, slugs can hold large quantities of liquid, which enhances the petroleum extraction rate. Slug flows also experience inherent fluctuations in wall heat transfer due to the varying rates of mixing and the presence of more than one phase with different fluid properties.

A large number of experimental and numerical studies have examined various aspects of gas-liquid slug flows. The majority of numerical studies have considered Taylor bubbles in laminar flows. The findings of these studies may not be applicable to typical oil, gas and nuclear systems, in which slug flows are usually turbulent. Another important issue that remains poorly understood is the interaction between adjacent Taylor bubbles [3], which affects bubble spacing and length, and, in turn, slug velocity, pressure drop, heat transfer, mixing, and pipe vibrations.

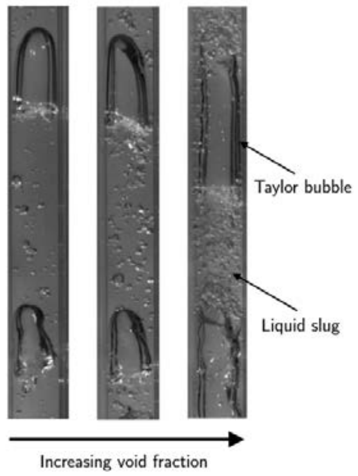

Fig. 1. Taylor bubbles rising in vertical channels.

The objective of the present study is to investigate numerically turbulent gas-liquid slug flows, in particular Taylor bubbles with turbulent wakes rising in stagnant water. In addition to examining the flow field around a single Taylor bubble, we also studied the interaction between two consecutive Taylor bubbles. Besides the velocity field, we also examined heat transfer around a Taylor bubble by heating passively the pipe wall.

2. Methodology

The computational model was constructed to match, as much as possible, the experimental setup in the study by Van Hout et al. [7], which we used to validate our simulations. These authors reported Particle Image Velocimetry (PIV) measurements around Taylor bubbles, containing air and rising in a 25 mm diameter tube, filled with stagnant water; the measurements were taken at an axial location of 2000 mm from the tube inlet. To keep the computational requirements within achievable levels, we adopted the moving reference frame technique, which has been applied for Taylor bubble simulations by previous investigators [3,8]. The computational domain comprised a circular tube with an inner diameter $\boldsymbol{D}$ = 25 mm and a length $\boldsymbol{L}$ = 500 mm ($\boldsymbol{L}/\boldsymbol{D}$ = 20). This domain was set to move downwards with a velocity equal to the experimental Taylor bubble rise velocity. Thus, the bubble appeared to be stationary within the computational domain, while liquid flowed downwards. A sketch of the computational domain is shown in Figure 2.

We performed all simulations using the CFD open-source software OpenFOAM 2.3 [4], which is a collection of C++ libraries that solve sets of partial differential equations governing a variety of physical problems. In particular, we used *InterFOAM*, a two-phase incompressible flow solver in OpenFOAM, which solves the two-phase equations of flow on collocated grids using finite volume discretization. A thorough description of relevant theory was recently presented by Deshpande et al. [5]. The same authors also validated the InterFOAM solver using different kinematic tests and found that it could simulate accurately time-dependent, two-phase flow problems, even without the use of fine meshes. To simulate time-dependent, separated, turbulent flows, we used the Spalart-Allmaras Improved Delayed Detached Eddy Simulation (IDDES) model [6], as implemented in OpenFOAM. This model combines advantages of Large Eddy Simulations (LES) and solutions of the Reynolds Averaged Navier Stokes equations (RANS) and allowed the study of coherent structures in the wakes of the Taylor bubbles.

The no-slip boundary condition was applied at the moving walls, while an advective boundary condition was applied at the bottom outlet of the computational domain. Additionally, a passive temperature field was introduced by applying a relatively low, constant heat flux $\boldsymbol{q''}$ at the walls. For consistency with the experimental setup, we fixed the origin of our coordinate system at the tail of the Taylor bubble (Figure 2).

To initialize the simulations, a cylindrical subdomain with a length of 4$\boldsymbol{D}$, a diameter of 0.8$\boldsymbol{D}$ and filled with air was introduced in the domain, which otherwise contained water. Water was introduced at the top of the domain with a uniform downward velocity $\boldsymbol{U_B}$ and the walls of the tube were also given a uniform downward velocity of $\boldsymbol{U_B}$. After a short (4 s) simulation time, the air bubble assumed its final bullet-like shape and an iterative process was used to determine the value of $\boldsymbol{U_B}$ for which the bubble nose remained stationary in the axial direction. The final bubble length and velocity in the present simulations were, respectively, 3.4$\boldsymbol{D}$ and 0.170 m/s, which are only slightly different from the corresponding experimental values of 3.6 ± 0.3$\boldsymbol{D}$ and 0.174 m/s.

Two sets of simulations were performed. One with a single air bubble and a second one with two bubbles. Both sets of simulations were run for approximately five flow-through times, at which time statistical convergence of the main flow quantities was achieved. Then, the simulations were continued for another five flow-through times for averaging. Running on 192 processors, each set of simulations required approximately 6 months of computer time. In the following, we will only report results of the single-bubble simulations.

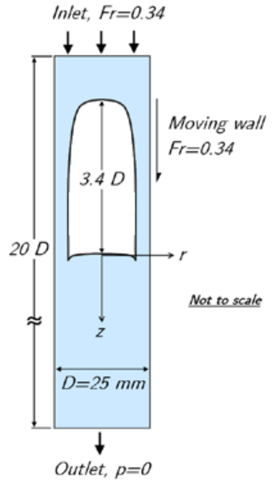

Fig. 2. Computational setup and boundary conditions; the Froude number is defined as $\boldsymbol{Fr = U_B/(gD)^{1/2}}$.

3. Results

3.1. Velocity field

As a Taylor bubble rises in a liquid column, liquid above it is displaced and flows downward in the annular region between the bubble and the tube wall. Because the thickness of the annular region is very small, the liquid in it moves with a relatively high velocity and exits the annular region as an annular jet, which forms a strong toroidal vortex in the immediate vicinity of the bubble bottom. This is the location were the most intense mixing is anticipated to occur. Our simulation results are consistent with these expectations (Figure 3). The wake of the bubble was found to be 2$\boldsymbol{D}$ long and the liquid jet detached from the tube wall at approximately $\boldsymbol{z}$=1.5$\boldsymbol{D}$, as a result of the observed adverse pressure gradient. Figure 4 shows that the predicted and measured radial profiles of the radial and axial velocity components at different axial locations downstream of the bubble were in very good agreement.

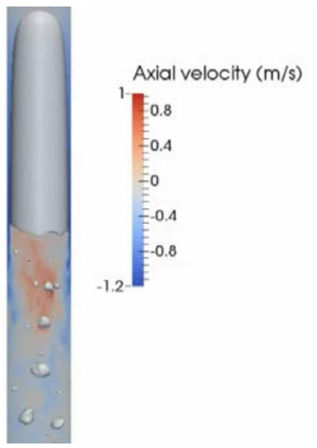

Fig. 3. Representative instantaneous velocity field around the Taylor bubble.

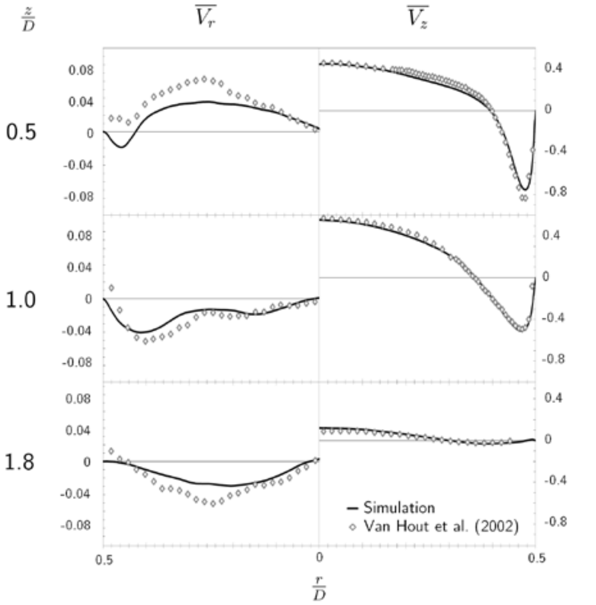

Fig. 4. Predicted and measured radial profiles of the axial and radial velocity components at three distances from the bubble bottom.

3.2. Temperature field

Figure 5 shows the axial variations of the non-dimensional wall temperature ***T**** and the Nusselt number **Nu** along the wall. Near the top of the domain, ***T**** was small and **Nu** was very large, due to the absence of a thermal boundary layer. As this boundary layer developed, ***T**** increased sharply and **Nu** dropped sharply. Near the Taylor bubble nose, the liquid accelerated as it entered the annular liquid film region and **Nu** increased gradually, while the wall temperature dropped accordingly. Near the Taylor bubble bottom, **Nu** increased again, following an increase in liquid velocity in the annular region, which reached values as large as $5U_B$. An inflection point in the axial profile of **Nu** is observed at the location where the liquid jet exited the annular region, below which the liquid decelerated. **Nu** peaked at an axial distance of approximately 1.5***D*** from the Taylor bubble bottom. This location was found to correspond to the separation point of the liquid film, where the velocity gradient vanished. This is also the location of minimum wall temperature. Downstream of this location, **Nu** gradually decreased and ***T**** gradually increased.

These observations have two main implications for the design and selection of materials for heat transfer systems that may experience slug flow. First, it is clear that a slug flow would be accompanied by thermal cycling; in the case of a constant wall heat flux, wall temperature at a certain location would tend to decrease while a Taylor bubble passed by and increase again downstream of the bubble, while a liquid slug was present. The frequency and magnitude of these fluctuations depend on the length of the Taylor bubbles, flow velocities and fluid properties. Secondly, to increase the overall heat transfer in slug flows, it is obviously desirable to have long Taylor bubbles and wakes. Wake length is relatively independent of Taylor bubble length, but Taylor bubble length can be controlled to some extent by influencing the process of Taylor bubble coalescence. In the next phase of this study, we will investigate this process and attempt to identify some of the parameters that control it.

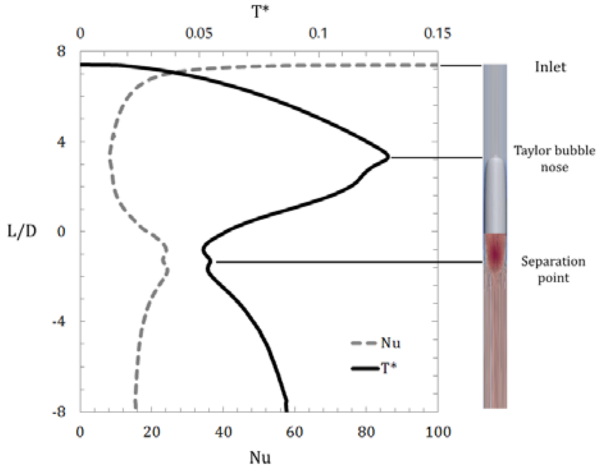

Fig. 5. Axial variations of the non-dimensional wall temperature $T^* = k(T_{wall} - T_{in})/(q''D)$ and the Nusselt number $Nu = (q''D)/k(T_{wall} - T_{bulk})$; k is the thermal conductivity of the liquid phase.

4. Summary

A numerical study of a Taylor bubble rising in a stagnant water column has been presented. The numerical results for the velocity field were validated vs. previous experiments and the temperature field produced by heating the wall was studied. It was observed that the convective heat transfer coefficient increased gradually in the liquid film surrounding the Taylor bubble and reached a maximum at the point of separation of the liquid film from the pipe wall, beyond which it decreased again. The wall temperature underwent large fluctuations during the passage of a bubble.

Acknowledgement: Financial support was provided by the University Network of Excellence in Nuclear Engineering (UNENE), Canadian Nuclear Laboratories and the Natural Sciences and Engineering Research Council of Canada (NSERC).

References

[1] Jones, O., Zuber, N., 1975. The interrelation between void fraction fluctuations and flow patterns in two-phase flow. Int. J. Multiphase Flow 2, 273-306.

[2] Taitel, Y., Bornea, D., Dukler, A., 1980. Modelling flow pattern transitions for steady upward gas-liquid flow in vertical tubes, AIChe Journal 26, 345-354.

[3] Lu, X., Prosperetti, A., 2009. A numerical study of Taylor bubbles. Ind. Eng. Chem. Res. 48, 242-252.

[4] The OpenFOAM Foundation, 2014. OpenFOAM 2.3, www.openfoam.com.

[5] Deshpande, S., Anumolu, L., Trujillo, M., 2012. Evaluating the performance of the two-phase flow solver Interfoam. Comput. Sci. Discovery 5, 014016 (37 pp.).

[6] Spalart, P. Deck, S., Shur, M., Squires, K., Strelets, M., Travin, A., 2006. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theor. Comp. Fluid Dyn. 20, 181-195.

[7] Van Hout, R., Gulitski, A., Barnea, D., Shemer, L., 2002. Experimental investigation of the velocity field induced by a Taylor bubble rising in stagnant water. Int. J. Multiphase Flow 28, 579-596.

[8] Taha, T., Cui, Z., 2006. CFD modelling of slug flow in vertical tubes. Chem. Eng. Sci. 61, 676-687.