Multiphase Flow of Taylor Bubbles in Rectangular Microchannels.

Regner, Dominik
2021

Multiphase Flow of Taylor Bubbles in Rectangular Microchannels

Student: Ing. Dominik Regner
Supervisor: Ing. Jaroslav Tihon, CSc.

The behavior of the gas-liquid flow in microchannels plays a crucial role in various industrial processes (*e.g.* compact heat exchangers, microreactors, or membrane modules). Despite the recent interest in CFD simulations of the Taylor flow, only fa ew studies have considered microchannels with cross-sections other than square or circular. Furthermore, the area of 2D simplification versus the width of the channel has not been explored in depth. The aim of this work is to broaden our knowledge of the behavior of Taylor bubbles in a coflowing liquid inside flat microchannels with a high aspect ratio. The primary focus is put on the near-wall region, which is difficult to study both experimentally or numerically. The investigated quantities are the thickness of a liquid film separating the bubble from the microchannel wall, wall shear stress induced by the bubble movement, and the shape and velocity of the translating bubble.

Numerical simulations are performed using software ANSYS Fluent and the volume of fluid method (VOF). The behavior of a large bubble (volume of 50 µl) carried by the liquid inside the microchannel with a rectangular cross-section (0.8 × 10 mm) is investigated. This configuration corresponds to the previous experiments by Tihon et al.[1] in which the electrodiffusion method is used to obtain the wall shear stress distribution over the microchannel and a high-speed camera to get the velocity and shape of the bubble. The calculations are carried out for small capillary numbers (Ca=0.0003-0.002) and Reynolds numbers corresponding to the laminar flow (Re=30-250). The CFD calculations follow the two methods used by Rocha et al.[2] and Magnini and Matar,[3] respectively. The first method is based on the relativity of motion with a bubble established in the domain together with liquid and the channel moving around it. The main advantages of this method are the small size of the computational mesh and the steadiness of the bubble. The main disadvantage is the necessity to search iteratively for the bubble velocity. The second method considers the bubble passing through the channel.

The results of CFD simulations are compared with theoretical predictions of the liquid film thickness by Bretheton,[4] and Han and Shikazono[5] and with the experimental results on the bubble velocity and wall shear stress by Tihon and al.[1] Further experiments are planned to validate CFD results as well as to elucidate the impact of simplification from 3D reality to 2D simulation.

References

1. Tihon, J.; Penkavova, V.; Stanovsky, P.; Vejrazka, J. Electrodiffusion Method of Near-Wall Flow Diagnostics in Microfluidic Systems. *Efm14 - Experimental Fluid Mechanics 2014* [Internet]. **2015**, 92.
2. Rocha, L.A.M.; Miranda, J.M.; Campos, J.B.L.M. Wide Range Simulation Study of Taylor Bubbles in Circular Milli and Microchannels. *Micromachines* **2017**, *8*, 154–173.
3. Magnini, M.; Matar, O.K. Morphology of long gas bubbles propagating in square capillaries. *Int. J. Multiphase Flow* **2020**, *129*, 103353.
4. Bretherton, F.P. The motion of long bubbles in tubes. *J. Fluid Mech.* **1961** *10*, 166–188.
5. Han, Y.; Shikazono, N. Measurement of liquid film thickness in micro-square channel. *Int. J. Multiphase Flow* **2009**, *30*, 842–853.