Proceedings of the 5th World Congress on Momentum, Heat and Mass Transfer (MHMT'20)
~~Lisbon, Portugal~~ Virtual Conference – October, 2020
Paper No. ICMFHT 189
DOI: 10.11159/icmfht.189

Numerical Simulation of Taylor Flow in the Entrance Region of Microchannels

Amin Etminan, Yuri S. Muzychka, Kevin Pope
Faculty of Engineering and Applied Science, Memorial University of Newfoundland
St. John's, NL A1B 3X5, Canada
aetminan@mun.ca; yurim@mun.ca; kpope@mun.ca

***Abstract* -** Computational Fluid Dynamics (CFD) has been widely employed by investigators to simulate Taylor flow in microchannels. High-resolution images captured by numerical techniques reveal significant details of an ultra-thin liquid film around the gas bubble. The interface between the liquid and the gas phases is a decisive factor in order to determine the flow pattern, the gas bubble profile, the bubble length–separation distance, the slug length, and so forth. Since the thickness of the liquid film is on the order of 10-15 μm for low capillary number flows, the mesh generation requires careful modeling to capture it and transport phenomena, such as momentum and heat. The present study is to develop a model of a transient Taylor flow in a two-dimensional microchannel using commercial ANSYS Fluent software. The Volume of Fluid (VoF) method was employed to simulate the interface between two phases, i.e., air and water. A comprehensive grid study was carried out to identify a sufficiently fine mesh to capture the film layer around the gas bubble in the uniform film thickness region. The curvatures of the nose and the tail of the gas bubbles were studied during traveling through the channel. Variations of the lengths of slugs and plugs were also measured as the functions of time. The numerical results were validated by available correlations of liquid film thickness, and slug length in previous studies.

***Keywords*:** Gas-Liquid; Microchannel; Numerical Simulation; Slug Length; Taylor Flow; Two-Phase Flow.