International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 04 | Apr -2017 www.irjet.net p-ISSN: 2395-0072

A review on analysis of air water two phase flow in conventional channel

Jagat B. Shihori[1], Krishna Kumar[2]

[1]PG Student, Department of Mechanical Engineering, SVMIT College Bharuch, Gujarat, India
[2]Assistant Professor, Department of Mechanical Engineering, SVMIT College Bharuch, Gujarat, India

---

Abstract – Two-phase flow has a wide range of engineering applications such as in the chemical & petroleum industries, refrigerators, micro reactors, ultra filtration plants. The hydrodynamic characteristics of the flow plays a very important role in the micro reactor design. generally the laminar flow occurs in conventional channel, so to achieve higher mass transfer large radial diffusion required to achieve. Radial mixing can be achieved in multiphase micro reactors. In this work, the hydrodynamic behavior of a air-water flow in capillary circular conventional-channel studying numerically and experimentally. Taylor flow pattern is main pattern occurring in two phase flow regime. It is most important flow regime to achieve higher mass transfer and heat transfer. Taylor flow enhances gas–liquid mass transfer. it has been observed that Taylor flow hydrodynamics and mass transfer performance are slug length and bubble velocity dependent. To study the effect of flow & geometrical properties on slug length and bubble velocity in taylor flow is important characteristic.

Key Words: **Air water two phase flow, conventional channel, Taylor bubble flow, two phase flow, slug flow**

1. INTRODUCTION

Two-phase flow main characteristic is the presence of interfaces, due to which the discontinuities presents in the properties over the interfaces and due to interface it can take number of shapes based on geometric configuration, inlet condition and superficial velocity of phases. The accurate measurement of two-phase flow parameters has always been a key parameter. Different flow regimes in two phase flow by variation of the gas–liquid superficial velocity ratios. Difference In horizontal and vertical channel orientation is effect of gravity on flows.in case of horizontal flow gravity acts perpendicular to flow. In vertical orientation, It acts opposite to flow direction if flow is upward direction and it acts in supporting direction if flow is downward direction. Thus channel orientation effects the flow regime greatly .mass transfer, pressure drop largely depends on particular flow pattern. Taylor flow pattern is very important flow patterns in different flow pattern occurring in two phase flow and because great mass transfer can be achieved in this flow pattern. Taylor flow is characterized by gas bubbles that almost fill the channel, separated by liquid slugs. The gas slugs are separation occurs by liquid slugs. Small liquid film connects the liquid slugs. The primary advantages offered by Taylor flow are the greatly reduced axial and improved radial mixing which can augment two or three phase reactions or enhance gas–liquid mixing. Taylor flow provides larger contact area between gas and liquid slugs which improves mass transfer. High surface-area-to-volume ratio increases its importance in design of heat exchangers and micro reactors.

Due to complex nature of Taylor flow regime, it has required more computation effort than other flow pattern.

1.1 GOVERNING EQUATIONS

The governing equations of the on multi-phase stream are recorded beneath.

Equation of continuity

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

Volume fraction equation

$$\frac{1}{\rho_p}\left[\frac{\partial}{\partial t}(\alpha_p \rho_p) + \nabla \cdot \alpha_p \rho_p \vec{v} = S + \sum_{q=1}^{n}(\dot{m}_{qp} - \dot{m}_{pq})\right]$$

Momentum equation

$$\frac{\partial(\rho\vec{v})}{\partial x} + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot \left[\mu(\nabla\vec{v} + \nabla\overrightarrow{v^{T}})\right] + \rho\vec{g} + \vec{F}$$

1.2 Volume of Fluid (VOF) Model

In the present study, VOF method is chosen to simulate two-phase flow. Simulation carried out with unsteady approach all simulations. The VOF model is the major multiphase model used in commercial CFD packages to capture interface. In VOF method, the interface is tracked by the solution of a continuity equation for the volume fraction of phases. The interface is tracked by geometric reconstruction scheme using the piecewise linear.

2 LITERATURE SURVEY

Tejas J. Bhatelia et al.[1] investigated the hydrodynamic behavior of gas-liquid system in flow capillary micro-channel using Computational Fluid Dynamics. They have modeled Taylor slug flow regime by volume of fluids (VOF) method. Different inlet geometries and channel diameters (2mm, 1mm and 0.5mm) are simulated. They selected equiangular (120º, inverted Y-junction) micro-channel to investigate the slug properties at various fluid and operating conditions. They performed simulation on both 2D and 3D inverted vertical Y-junction channel and compared the results obtained. The effect of gravity, superficial velocities of both the phases and contact angle is studied. Based on results, it can be concluded that surface tension affects the slug characteristic.

Raghvendra Gupta et al.[2] have studied the two-phase Taylor flow in a conventional channel using ANSYS fluent. the gas–liquid interface is tracked by geometric reconstruction scheme. It represent the interface by piecewise-linear approach. Air and water were used as the working fluids. They have studied the effect of wall adhesion and surface tension effect on two phase flow and observed that wall adhesion and surface tension affects the slug lengths. They have observed that gas and liquid slug length depends on channel inlet condition.

Dongying Qian, Adeniyi Lawal[3] studied two-phase flow through a T-junction empty channel. The channels selected had varying sizes (0.25, 0.5, 0.75, 1, 2 and 3 mm). They have used commercial computational fluid dynamics (CFD) package, FLUENT for the numerical simulation. Interface is tracked in FLUENT by the volume of fluid (VOF) model. The gas and liquid slug length studied at different inlet conditions, at different gas and liquid superficial velocity, at different surface tension & liquid viscosity and contact angle. The numerical results found in good agreement with the experimental results. They have observed gas and liquid slug lengths depends on gas and liquid superficial velocity. By increasing the gas velocity, gas slug length start increase at constant liquid velocity. By increasing the liquid velocity, gas slug length start decreases at constant gas velocity .They have given the bubble velocity equation which equal to summation of gas and liquid superficial velocity.

G. Rosengarten, D.J.E. Harvie, J. Cooper-White[4] the effect of contact angle on liquid droplet formation through a contraction. They have studied the effect of contraction on droplet formation using the FLUENT. They have concluded that droplet size and shape greatly effected by the wall contact angle.

Bin Chen, Guo Fang [5] studied the the Taylor bubble flow inside the conventional channel using the FLUENT . They have generated the taylor flow at particular superficial. They have observed that gas and liquid slug lengths are superficial velocity dependent and it also depends on channel orientation. They have concluded that two regimes observed based on capillary sizes. At lower capillary no surface tension force dominant and at higher capillary number shear force dominant over surface tension and small slugs are formed. They have observed when Ca number is lower surface tension effect is larger and shear effect can be neglected and due to higher surface tension larger slug observed. the results are also visualized experimentally and good agreement observed

Taha Taha, Z.F. Cui[6] investigated the taylor bubble motion in conventional channel using the volume of fluid method in FLUENT. They have captured the motion of single gas bubble. Taylor bubble has spherical nose and variation observed in taylor bubble tail shape. they have explained the mechanism of highly agitated mixing in taylor flow.it is observed that as the taylor bubble velocity increases the velocity of falling liquid film also start increases and falling liquid film merge with liquid slug and creates the mixing zone.

M. K. Akbar, S. M. Ghiaasiaan[7] carried out analysis of air water two phase in taylor flow regime using the ansys FLUENT. They have created geometry using the GAMBIT software and meshing done and further analysis carried out by exporting file in FLUENT. They have studied the slug length, gas holdup and bubble velocity for range of gas and liquid inlet velocity. They have carried out simulation for both horizontal and vertical channel orientation and observed that in both channel orientation as two phase velocity increases bubble velocity get increases. Bubble velocity results are compared with experimental result and good agreement found between simulation and experimental result.

W. Salman, A. Gavriilidis, P. Angeli[8] studied the taylor bubble generation inside channel. they have modelled bubble generation using the FLUENT. They have identified that bubble formation process passes through three stage process .first any bubble generated get expanded then contraction occur and finally necking of bubble occurs. Bubble shape and size depends on first two stages. Different bubble shape and size observed for different channel size and orientation.

Yuming Chan et al.[9] carried out the numerical simulations for taylor flow regime. They have carried out analysis on bubble size and shape by varying the liquid and gas velocity. They have observed that bubble shape and size get affected by liquid and gas superficial velocity. They have also carried out simulation by changing inlet geometry and keeping other parameters constant and concluded that bubble shape and size greatly depend on inlet geometry.

Donghong Zheng, Xiao He, Defu Che[10] simulated the taylor flow using the FLUENT. They have observed the effect of surface tension, viscous force effect on taylor bubble characteristic .they have captured the liquid film surrounding the taylor bubble by adopting the fine mesh. They have concluded that liquid film always surround the gas bubble in case of fully wetting characteristic fluid.

Hui Liu at al.[11] experimentally studied air water two phase flow in conventional channel they have obtained taylor bubble flow in three different water, ethanol, and an oil mixture, to study the effect of liquid properties on bubble properties. They have calculated the bubble velocity experimentally. They have observed that bubble velocity is greatly affected by fluid properties such as surface tension, viscous effect. Phase hold up is also studied. Gas hold up increases with increase in gas superficial velocity. They have obtained the correlation for bubble velocity as mentioned below

$$\frac{V_b}{U_{tp}} = \frac{1}{1 - 0.61Ca^{0.33}}$$

Where V_b is bubble velocity and U_{tp} is two phase velocity,Ca is capillary number.

S. Laborie, C. Cabassud et al.[12] have experimentally studied slug flow in small diameter capillaries (up to 4 mm). Experiments showed that bubble moves faster than mean velocity flow due to slip between the bubble and falling liquid film. They have compared the bubble velocity for different fluid for concluding the effect of fluid properties on bubble velocity. They have observed that gas slug increases with increase in gas superficial velocity for same liquid velocity. They have compared slug length in case of air-water which has surface tension very high compared to air-ethyl alcohol and slug length found higher in case of air–water system which confirms that surface tension greatly effects slug length.

Hemant B. Mehta, Jyotirmay Banerjee[13] carried out the experimental investigations on taylor bubble flow in channel. Based on experiment , they have calculated the taylor bubble velocity. Taylor bubble length increases with increase in gas superficial velocity and becomes maximum with lower liquid superficial velocity. Gas hold up also increases with gas superficial and Liquid slug length on the other hand decreases with increase in gas superficial velocity and increases with increase in liquid superficial velocity.

3. CONCLUSIONS

Review of taylor bubble flow pattern of air water two phase inside conventional channel is carried out numerically as well as experimentally. Based on review, It is concluded that
•Hydrodynamic characteristic of taylor bubble is greatly affected by gas and liquid superficial velocity, inlet channel geometric configuration, channel orientation.
•The liquid fluid properties affects the characteristic of taylor bubble.it is observed that gas slug length and size gets increases with increase in surface tension.

ACKNOWLEDGEMENT

I express my gratitude to my guide Prof. Krishna kumar for his expert guidance, encouragement and suggestion.

REFERENCES

[1] Bhatelia, T. J., Pareek, V. K., Utikar, R.P., Tade M.O., (2009), "Hydrodynamics of slug flow in capillary microchannels", University journal of Engineering and Technology 1, Pages 1-9.

[2] Raghvendra Gupta, David F. Fletcher, Brian S. Haynes (2009), "On the CFD modeling of Taylor flow in microchannels", Chemical Engineering Science (64), Pages 2941-2950.

[3] Qian D., Lawal A., (2006), "Numerical study on gas and liquid slugs for Taylor flow in a T-junction micro-channel", Chemical Engineering Science (61), Pages 7609-7625

[4] G. Rosengarten, D.J.E. Harvie, J. Cooper-White. (2006), "Contact angle effects on micro-droplet deformation using CFD", Applied Mathematical Modelling, Volume 30, Issue 10, Pages 1033-1042.

[5] Bin Chen, Guo Fang, (2009), "Numerical study on Taylor bubble formation in a microchannel T-junction using VOF method", Microgravity Science Technology 21, Pages 51-58.

[6] Taha T., Cui Z.F., (2006a), "CFD modeling of slug flow in vertical tubes", Chemical Engineering Science (61), Pages 676–687.

[7] M. K. Akbar and S. M. Ghiaasiaan (2006), "Simulation of Taylor flow in capillaries based on the volume-of-fluid techniques", Industrial & Engineering Chemistry Fundamentals (45), Pages 5396–5403

[8] Salman W., Gavriilidis A., Angeli P., (2006), "On the formation of Taylor bubbles in small tubes", Chemical Engineering Science 61 (20), Pages 6653–6666.

[9] Yuming Chen, Rudi Kulenovic, Rainer Mertz, (2009), "Numerical study on the formation of Taylor bubbles in capillary tubes", International Journal of Thermal Sciences 48 (2), Pages 234–242.

[10] Donghong Zheng, Xiao He, Defu Che (2007), "CFD simulations of hydrodynamic characteristics in a gas–liquid vertical upward slug flow", International Communications of Heat and Mass Transfer, 50, Pages 4151-4165

[11] Hui Liu, Chippla O. Vandu, and Rajamani Krishna (2005), "Hydrodynamics of Taylor Flow in Vertical Capillaries: Flow Regimes, Bubble Rise Velocity, Liquid Slug Length, and Pressure Drop", Ind. Eng. Chem. Res., Vol. 44, Pages 14-28.

[12] S. Laborie, L. Cabassud and Durand-Bourlier (1999), "Characterisation of gas-liquid two phase flow inside capillaries", Chemical Engineering Science (54),

[13] Mehta H. B., Banerjee J., (2016), "Experimental investigation on thermo hydrodynamics of continuous Taylor bubble flow through mini channel ", International Journal of Heat and Mass Transfer (94), Pages 119–137

John W. Coleman, Srinivas Garimella, (1999), "Characterization of two phase flow patterns in small diameter round and rectangular tubes", International Journal of Heat and Mass Transfer ,Volume 41 , Pages 2869-2881.