IJE TRANSACTIONS C: Aspects Vol. 36, No. 03, (March 2023) 540-547

International Journal of Engineering

Journal Homepage: www.ije.ir

Numerical Analysis on Flow Characteristics of Gas-liquid Two-phase Flow in a Vertical Pipe with Downward Stream

S. Hasanpour, I. Mirzaee, M. Khalilian*

Mechanical Engineering Department, Faculty of Engineering, Urmia University, Urmia, Iran

PAPER INFO

Paper history:
Received 19 November 2022
Received in revised form 17 December 2022
Accepted 24 December 2022

Keywords:
Two-phase Flow
Downward Flow
Numerical Simulation
Level Set Method

ABSTRACT

Through this paper, a 3D simulation together with experimental observation was conducted to study two-phase flow in a vertical tube. OpenFOAM software was employed to analyze air and water. Main flow stream was downward which was considered to be within a vertical pipe of 10 mm in diameter. Study included two inputs for flows: upper input for water and side input for air. Several states with various mass fluxes for both water and air were studied. Based on physics of the issue, numerical simulation was considered to be time-dependent. Obtained results showed that when air velocity occupied lower values, air momentum cannot overcome water momentum leading in small slugs. When airflow velocity was more than water flow rate, it dominated water flow and consequently could affect mainstream direction. Also, velocity graphs on centerline represented that going forward in time, velocity magnitude experiences a significant value of fluctuations and large oscillations occur next to outlet. Comparing experimental and numerical results, approximately 9% differences can be found which showed suitable agreement. Results showed that at initial steps, void fraction faces a significant jump in values. Intensity of this change in void fraction values was higher in lower water velocity. Indeed, by increment of water velocity, inertial forces associated with liquid phase find a dominant role in overall hydrodynamics of the gas-liquid flow. Also, it is obvious that flowing manner in cases 1, 2, and 3 are similar but after case 4, flow pattern varies. These changes are more considerable in cases 5 and 6.

***doi**: 10.5829/ije.2023.36.03c.14*

NOMENCLATURE

D	Pipe diameter, m
F	Force, kg m/s^2
g	Gravitational acceleration, m^2
h	Enthalpy
k	Turbulence kinetic energy, m^2/s^2
p	Static pressure, pa
Q	Volume flow rate, m^3/s
q	Generation term
T	Turbulent viscosity tensor
Re	Reynolds number
t	Time, s
u	Velocity, m/s

Greek Symbols	
α	Void fraction
ρ	Density, kg/m^3
μ	Dynamic viscosity, Pa.s
ω	Specific dissipation rate, kg/m^{2}s
Subscript	
i	Interaction
int	Initial
G,g	Gas phase
L,l	Liquid phase
c	Continues phase
d	Dispersed phase
k	Phase k
s	Surface

1. INTRODUCTION

The multiphase flow is named for the stream of two substances that are in different phases from the viewpoint of physics. Through the last decades, it was revealed that multiphase flows (especially the application is gas\liquid two-phase flow) could occur within a variety of industrial applications. The industrial applications that included the gas-liquid two-phase flow included power plants, fuel refinery systems, drug production processes, and so on.

---

*Corresponding Author Institutional Email: m.khalilian@urmia.ac.ir (M. Khalilian)

Please cite this article as: S. Hasanpour, I. Mirzaee, M. Khalilian, Numerical Analysis on Flow Characteristics of Gas-liquid Two-phase Flow in a Vertical Pipe with Downward Stream, ***International Journal of Engineering, Transactions C: Aspects,*** Vol. 36, No. 03, (2023) 540-547

These high technological applications urge the investigations to better understanding of the nature of the multiphase flows [1, 2].

A type of multiphase flow named the gas-liquid two-phase flow is one of the major types of multiphase flow that widely exists in natural and industrial applications. The complexity of the nature of these types of streams makes them very interesting for researchers of this field. This complexity is based on the nature of two different phases and could be intensified by the variations in the orientation, shape, and form of the flowing path. Findings have presented that the gas-liquid two-phase flow could present different thermal and hydraulic behavior in different flow patterns [3]. They represented that the centrifugal forces influence the flow pattern in helical tubes. Golan [4] implemented an experimental investigation within a 0.038 m diameter round tube that the air-water two-phase flow was considered as the working fluids. Through their investigation, the flow patterns of bubbly, slug, oscillatory, and annular flow patterns were observed. The hydrodynamic behavior of the flow around a Taylor ascending bubble using the VOF model was numerically studied by Taha and Cui [5]. The results represented that the movement of this bubble creates a very thin layer of film on the sides of the bubble and in contact with the wall. Hossain et al. [6] employed the VOF model to study the oscillating behavior of slug flow. They utilized the k-ε model for turbulence flow simulation. An empirical investigation of the mixing of gas-liquid flow within horizontal pipes was conducted by Akhlaghi et al. [7]. They performed a numerical analysis to compare the results with the experimental ones. Their results showed that when the surface velocity of the gas phase is enhanced at a constant surface velocity of the liquid, the fraction of the gas slug length to that of the liquid slug goes up significantly. Yu et al. [8] worked on the thermal and hydrodynamic features of slug flow in a microchannel. In their study, they also utilized the VOF model to simulate this flow. Gupta et al. [9] experimentally studied the thermal and hydrodynamic characteristics of two-phase flows including the Taylor bubble employing a fluid volume model. The results reported that the Nusselt number of two-phase flows is up to 2.5 times higher than single-phase. Newtonian and non-Newtonian fluids including two phases examined by Ratkovich et al. [10]. They provided a relationship to predict the volume fraction values. Zheng et al. [11] numerically investigated the features of the Taylor bubble in the ascending flow in the vertical tube. The results of their study represented that in the flow where the inertial force is predominant, the shear stress of the wall enhances with an increase in velocity of the Taylor bubble. Abdulkadir et al. [12] compared the experimental and numerical results of the Taylor bubble studying. The results of their study represented that the use of the VOF model can provide an acceptable result for studying the features of two-phase flow models.

An investigation conducted by Adegoke and Oyediran [13] included the nonlinear dynamics behavior of cantilevered pipes that included the gas-liquid two-phase flow. Through their studies, they applied different scales of methods to provide the axial and transverse vibrations. Furthermore, the influence of Poisson’s ratio and the pressurization on the dynamic response of the pipe were probed through their results. Wang et al. [14] represented a dynamic configuration of a horizontal pipe that consisted of slug flow. They applied the finite element method for solving the governing equation associated with the motion of gas liquid two phase flow. Their results presented a good agreement between the results of the simulation and the empirical results. Their results presented a significant impact of the slug transition velocity on the main features of the system as like to damping. Mimouni et al. [15] reported that the changes in the values of the flow rates of both phases in the inlet section of the horizontal pipes results in the presence of various flow patterns. They reported that the flow patterns may consist of slug, plug and intermittent flow patterns. Also, it was found that the intermitted nature of the gas-liquid flow interrupts the ordinary performance of the instruments associated with the oil and gas refinery systems [16] and may cause some very significant damages to the pipelines [17]. An investigation implemented by Montgomery [18] focused on the flow properties within an S-shaped riser. They probed the feaures of flow patterns associated with various separator pressures and developed a stability criterion to observe the stable and unstable flow. Malekzadeh et al. [19] performed some experiments associated with long pipeline-riser systems. They tried to focus in the features of the gas-liquid two-phase flow in these systems. Xie et al. [20] studied the influence of backpressure on severe slugging characteristics in a large flow loop with a total pipeline length of 405 m. Assari et al. [21] utilized the mixture method to simulate the gas-liquid two-phase flow inside an air-water ejector. Their results revealed a deviation of 7% between the simulation results and the existing empirical data. Han et al. [22] tried to propose and develop a new technology to determine the void fraction which was based on an artificial neural network.

Based on the literature reviewed, there are limited numerical works on the downward air-water two-phase flow in detail. Also, some papers showed the results without any comparison with experimental data which is done in this work. Therefore, herein, this kind of flow is studied numerically in a 3D vertical pipe through various mass fluxes. The present study includes several sections. The first section is an introduction and literature review of the proposed issue. The next part has essential definitions of numerical methodology. Then, the result and discussion part is discussed. Finally, the conclusion and studied references, specific parts are shown, respectively.

2. METHODOLOGY and VERIFICATION

The geometric configuration was developed and meshed in Gambit software. Then, the desired model is exported to OpenFOAM software and the simulation steps in this software are continued.

Six balance equations are defined in an overall two-fluid model. However, seven different dependent flow factors of α_g, $u_g . u_l . h_g . h_l . p_g$ and p_l play their roles which brings a kind of complexity to solving the mentioned equations. There are two individual methods to get a complete set of equations. The first is the definition of more simplifications to diminish the number of dependent parameters. The second one is to develop further differential equations to get an equilibrium condition between the number of dependent parameters and equations. Thus, usually, the assumption of equal local pressure $p_g = p_l = p$ are stated because the variation in local pressure among the individual phases was considered to be slight and could be considered negligible for various applications. Supposing a unique local pressure amount, the below mass, momentum, and energy conservation equations could be extracted:

Conservation of mass equation:

$$\frac{\partial}{\partial t}\alpha_k\rho_k + \nabla(\alpha_k\rho_k\boldsymbol{u}_k) = \Gamma_k \tag{1}$$

Momentum balance equation:

$$\frac{\partial}{\partial t}\alpha_k\rho_k\boldsymbol{u}_k + \nabla(\alpha_k\rho_k\boldsymbol{u}_k \otimes \boldsymbol{u}_k) + \alpha_k\nabla p + \left(p - p_k^{int}\right)\nabla\alpha_k = \alpha_k\rho_k g + F_k^{int} + \Gamma_k u^{int} - \nabla.(\alpha_k T_k) \tag{2}$$

Energy balance equation:

$$\frac{\partial}{\partial t}\alpha_k\rho_k h_k + \nabla(\alpha_k\rho_k h_k\boldsymbol{u}_k) - \alpha_k\frac{D_k p}{Dt} = \Gamma_k h_k^{int} + q_k^{ext} \tag{3}$$

Through which the term k equals g for gas or vapor. Furthermore, the k equals l for the liquid phase. $\frac{D_k}{Dt}$ stands for the material derivative and T_k stands for the turbulent viscosity tensor [23]. The main configuration is a three-dimensional pipe of 10 mm in diameter with a side input at the head. To do the numerical evaluation, three different mass fluxes of 0.75, 1.5, and 2 l/min were considered for the water flow rate. Simultaneously, five mass fluxes of 0.01, 0.2, 1, 3, and 5 l/min were considered for air flow rate. Table 1 shows the related velocities for all mass fluxes.

Level-Set Method (LSM) was employed to simulate the problem. The level-set method is a popular interface-tracking method for computing two-phase flows with topologically complex interfaces. In the level-set method, the interface is captured and tracked by the level-set function, defined as a signed distance from the interface. Assuming the curve of Γ is perpendicular to the speed of v, then the level-set function φ could easily satisfy the level-set equation:

TABLE 1. Various velocity values for water and airflow

Cases	Water vel. [m/s]	Air vel. [m/s]
Case 1 (based case)	0.01	0.005
Case 2	0.02	0.005
Case 3	0.01	0.008
Case 4	0.0265	0.053
Case 5	0.0265	0.159
Case 6	0.0265	0.265

$$\frac{\partial\varphi}{\partial t} = \mathrm{v}|\nabla\varphi| \tag{4}$$

The solution φ of Equation (4) describes the time-dependent position of the interface Γ(t), implicitly as its zero level, i.e.

$$\Gamma(t) = \{x\epsilon\Omega: \varphi(x,t) = 0\} \tag{5}$$

This approach renders the method robust with respect to topological changes of the interface. For instance, a collision of two droplets can be handled easily. Figure 1 shows the main flow domain. As it is observed, water is fed to the pipe from the head and the air enters from the side input. In both inlets, the velocity value is known as previously mentioned and the outlet consists of specified relative pressure equal to zero.

The mesh independency, herein, is the first step. Figure 2 shows that the unstructured meshes are used in this simulation.

In this study, firstly, five different cell numbers are investigated. To assess solution mesh independence, Figure 3 is extracted. This figure shows that the magnitude of outlet velocity varies by changing cell numbers. It is clear that after 350,000 cells, the changes

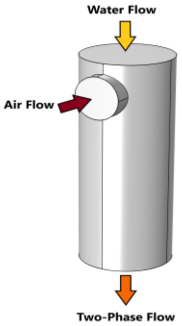


Figure 1. Main simulated domain

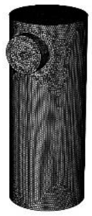


Figure 1. Unstructured used meshes

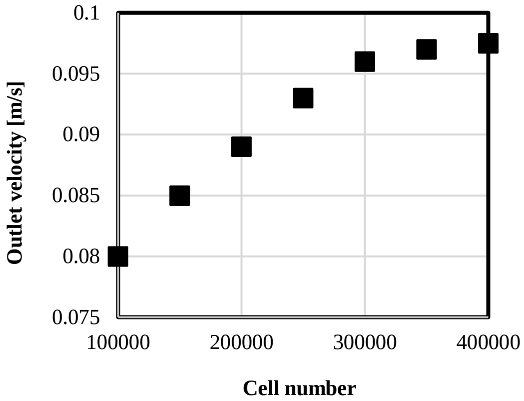


Figure 3. Solution mesh independence based on outlet velocity for 1s

in velocity magnetite is negligible. Therefore, to reduce computation cost, this number of cell is selected to fulfill the numerical procedure.

The most important part of the numerical investigations is the reliability checking of the results. To this aim and for the validation of the simulation results, the results of the flow map were compared with those obtained with Oshinowo [24]. Figure 4 depicts the flow map results of the present simulation in comparison with the results of Oshinowo [24]. Looking at the mentioned figure, it could be concluded that the flow transition points between the flow patterns of bubbly-slug, slug-froth, and froth-falling film of the simulation results are in good agreement with those associated with the results of Oshinowo [24]. Although there is a minor deviation between the results of the present simulation and the results of the Oshinowo [24], this difference can be explained based on the differences in the description of the physical structure of the flow patterns defined by individual investigators.

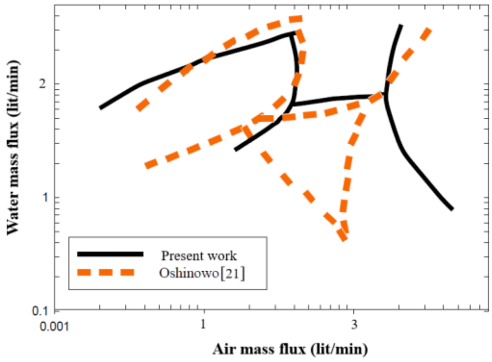


Figure 2. Flow pattern comparison for present work vs. Oshinowo [24].

3. EXPERIMENTAL SETUP

The setup consisted of two main streams including the airlines and waterline. For the water line, a tank was used as water storage and a drain was fabricated to exit the extra water. A pump supplied the water from this tank. It is worth to be mentioned that the tank was filled with tape water. A mixer was located in the next step that combined the airflow with water flow to generate the two-phase flow. It should be noted that the airflow was produced by employing a compressor with suitable pressure to feed the mixer. Furthermore, three individual rotameters (KHL-08A01M-V mode) were utilized to sense the volumetric flow rates of each stream. The test section included a 4 m pipeline through which a camera was located in the last 1 m for capturing the flow structure. A schematic depiction of the experimental setup is shown in Figure 5.

4. RESULTS and DISCUSSION

Firstly, according to Table 1, case 1 is the base case for this simulation. Thus massive data are extracted for this case to show the flow dynamics in the investigated domain then the results of other cases were compared to each other briefly. The previous findings have revealed that the void fraction is a function of some different factors as the mass flow rate of the individual mass flow rates of each phase and also the flow patterns. Through the dominant flow patterns that are observed in

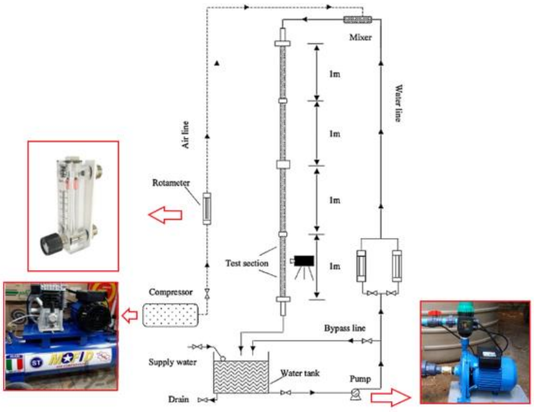


Figure 5. Schematic of the experimental setup

downward flows the minimum values of void fraction were found to be associated with the bubbly flow, whereas the maximum values were always relevant to the annular flow patterns. Through Figure 6, the variation of void fraction in terms of the variation of air flow rate and for different water flow rates are depicted. It was found that at the initial steps the void fraction faces a significant jump in the values. The intensity of this rapid change in the values of the void fraction was higher in the lower values of the water velocity. Indeed, by the increment of water velocity, the inertial forces associated with the liquid phase find a dominant role in the overall hydrodynamics of the gas-liquid two-phase flow. Looking at Figure 6, it could be concluded that after a certain value of the air velocity the void fraction curves face an almost constant slope. It should be noted that the above-mentioned statement is in good agreement with previously presented findings of some other investigations, from which their work were published by Jiang and Rezkallah [25], Usui and Sato [26], Nguyen [27] and Vatani and Domiri-Ganji [28] could be named.

Since the solution domain is 3D so to deduce understandable results, it is essential to specify a certain line to extract data on which. Figure 7 shows this centerline. Also, it is worth mentioning that to reduce solution time, the simulation is run out up to 5s.

The first parameter studied on the centerline is velocity magnitude. Figure 8 represents this parameter for different periods. This figure shows that by going forward in time, the centerline velocity oscillation is larger. Furthermore, it is obvious that near the outlet, the fluctuations bigger. According to the geometry and base case parameters, water flow is dominant initially while time spending, and air volume fraction increase, and these phenomenon cause fluctuations in the mainstream. Figures 9 and 10 demonstrate such facts.

Figure 11 shows velocity contours to clarify the happened physical phenomena in the studied domain.

Figure 12 is a comparison among velocity contours for all cases at plane x=0.0 mm in 5s. As shown, when air velocity is less, airflow cannot overcome the mainstream of water while when air has a higher velocity value, it can dominate water momentum and can cause direction change for the mainstream. To show slug or bubble flow, it is recommended to represent the air volume fraction on the XZ plane. Therefore, Figure 13 is extracted.

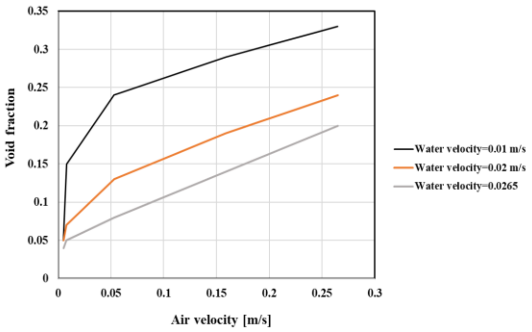


Figure 6. The void fraction for different water and air velocities

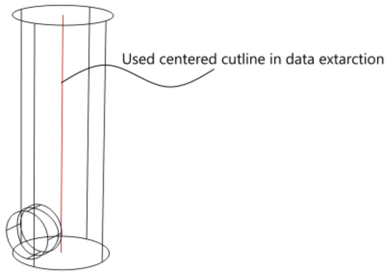


Figure 7. Specified centerline in the studied domain

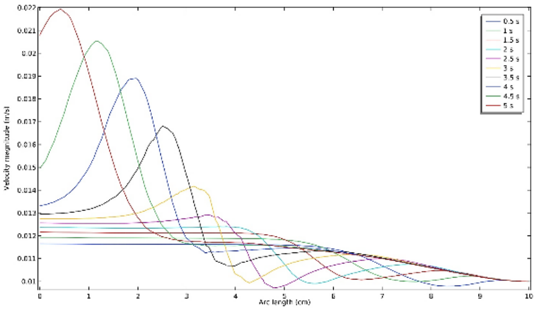


Figure 8. Velocity magnitude for base case on the centerline via time

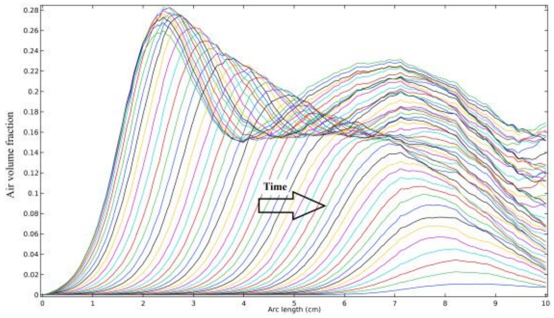


Figure 9. Air volume fraction during the time for the base case

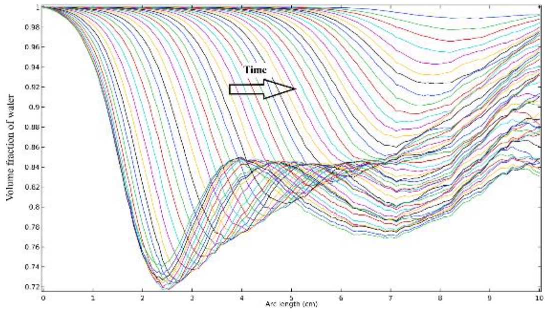
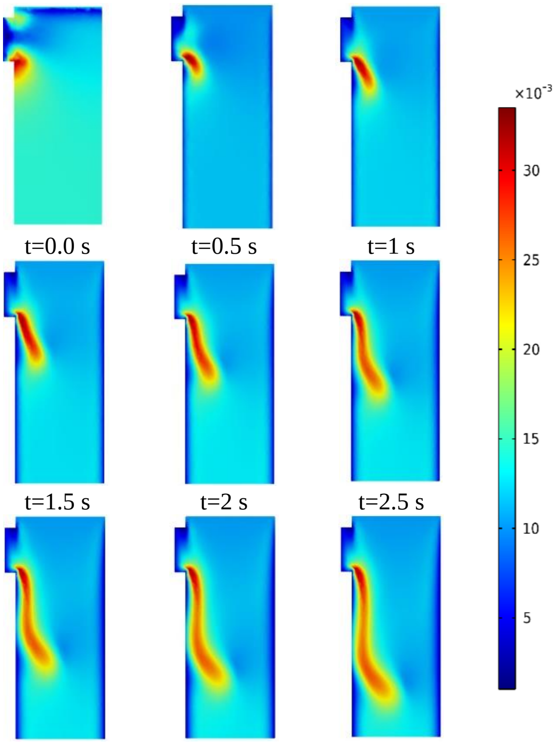


Figure 10. Water volume fraction during the time for the base case

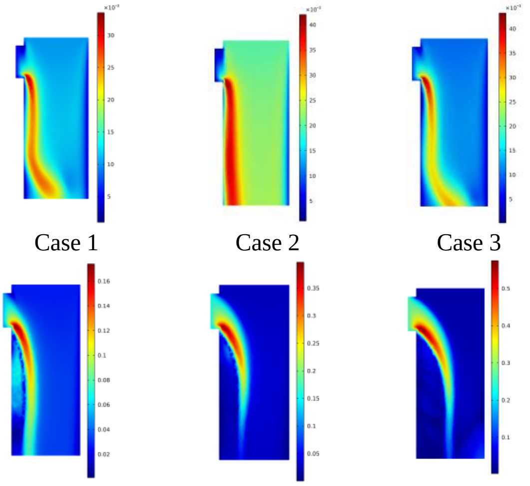


t=0.0 s t=0.5 s t=1 s

t=1.5 s t=2 s t=2.5 s

t=3 s t=3.5 s t=4 s

Figure 11. Velocity contours for the base case at x=0.0 mm



Case 1 Case 2 Case 3

Case 4 Case 5 Case 6

Figure 12. Comparison of velocity contours for all cases in 5s at x=0.0

Figure 13 is the 3D void fraction for all cases. It is clear that the flowing manner in cases 1, 2, and 3 are somewhat similar but after case 4, the flow pattern varies. These changes are more considerable in cases 5 and 6. In fact, with specified water influx and increased air flow rate this happens. To compare the numerical results with the achieved experimental ones, Figure 14 is extracted as follows:

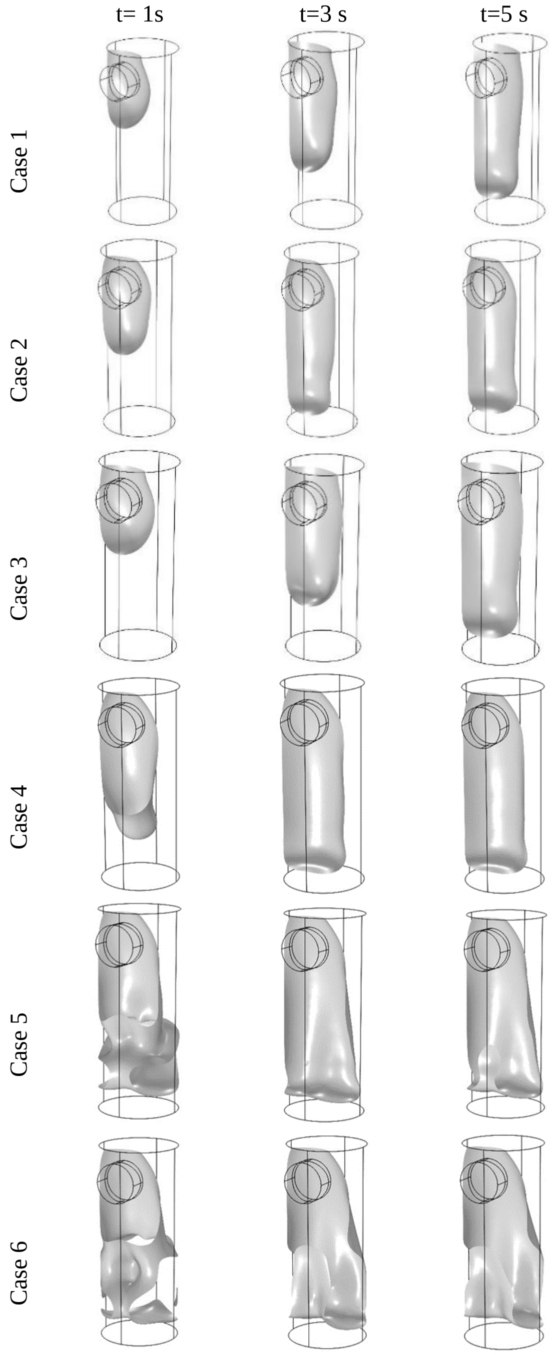
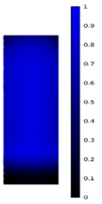
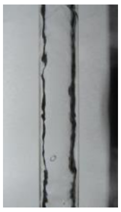


Figure 13. 3D void fraction for All cases at 1, 3, and 5s

Water mass flux/Air mass flux	**Numerical**	**Experimental**
0.5		

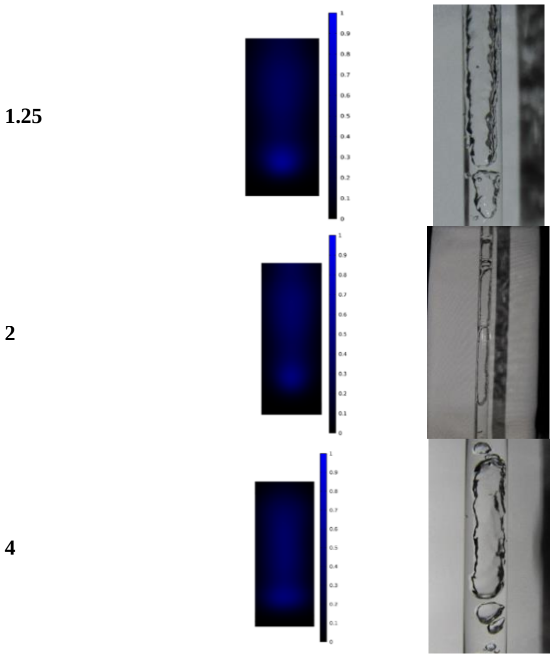


Figure 14. Comparison of experimental and numerical results for various ratios of water and air mass flux

5. CONCLUSION

The present work included a numerical simulation and an experimental study of downward gas-liquid two-phase flows within a pipe with 10 mm of diameter. The main domain consisted of two inputs for flows. The head input was for water as the liquid phase and the side input was for air as the gaseous phase. In the first step of the numerical investigation, solution mesh independence is investigated then six cases with different mass fluxes were studied. The simulation was transient modeling. To reduce the computation time, just 5 seconds was considered for extracting data. The level set method was utilized to model two-phase interactions. The verification process showed there was a minor deviation between the results of the present simulation and the results of the reported data by others. The achieved results demonstrated that when airflow velocity was low, air momentum could not overcome water momentum so small slugs or massive bubbles were observed but when airflow velocity is more than water one, it can dominate water flow and consequently can affect the mainstream direction. In such cases actually, airflow is dominant and covers almost all core of the pipe. Comparing experimental and numerical results, approximately 9% differences can be found so it shows suitable agreement.

6. REFERENCES

1. Jafarmadar, S., "The effects of pressure difference in nozzle's two phase flow on the quality of exhaust mixture", ***International Journal of Engineering, Transactions B: Applications***, Vol. 26, No. 5, (2013), 553-562. doi: 10.5829/idosi.ije.2013.26.05b.12.
2. Tayari, E., Torkzadeh, L., Domiri Ganji, D. and Nouri, K., "Analytical solution of electromagnetic force on nanofluid flow with brownian motion effects between parallel disks", ***International Journal of Engineering, Transactions B: Applications***, Vol. 35, No. 8, (2022), 1651-1661. doi: 10.5829/IJE.2022.35.08B.21.
3. Murai, Y., Inaba, K., Takeda, Y. and Yamamoto, F., "Backlight imaging tomography for slug flows in straight and helical tubes", ***Flow Measurement and Instrumentation***, Vol. 18, No. 5-6, (2007), 223-229. https://doi.org/10.1016/j.flowmeasinst.2007.07.001
4. Golan, L.P., "An air-water study of vertical upward and downward two-phase flow", (1969).
5. Taha, T. and Cui, Z.F., "Cfd modelling of slug flow in vertical tubes", ***Chemical Engineering Science***, Vol. 61, No. 2, (2006), 676-687. https://doi.org/10.1016/j.ces.2005.07.022
6. Hossain, M., Chinenye-Kanu, N.M., Droubi, G.M. and Islam, S.Z., "Investigation of slug-churn flow induced transient excitation forces at pipe bend", ***Journal of Fluids and Structures***, Vol. 91, (2019), 102733. https://doi.org/10.1016/j.jfluidstructs.2019.102733
7. Akhlaghi, M., Mohammadi, V., Nouri, N.M., Taherkhani, M. and Karimi, M., "Multi-fluid vof model assessment to simulate the horizontal air–water intermittent flow", ***Chemical Engineering Research and Design***, Vol. 152, (2019), 48-59. https://doi.org/10.1016/j.cherd.2019.09.031
8. Yue, C., Zhang, Q., Zhai, Z. and Ling, L., "Cfd simulation on the heat transfer and flow characteristics of a microchannel separate heat pipe under different filling ratios", ***Applied Thermal Engineering***, Vol. 139, (2018), 25-34. https://doi.org/10.1016/j.applthermaleng.2018.01.011
9. Gupta, R., Fletcher, D.F. and Haynes, B.S., "Cfd modelling of flow and heat transfer in the taylor flow regime", ***Chemical Engineering Science***, Vol. 65, No. 6, (2010), 2094-2107. https://doi.org/10.1016/j.ces.2009.12.008
10. Ratkovich, N., Majumder, S. and Bentzen, T.R., "Empirical correlations and cfd simulations of vertical two-phase gas–liquid (newtonian and non-newtonian) slug flow compared against experimental data of void fraction", ***Chemical Engineering Research and Design***, Vol. 91, No. 6, (2013), 988-998. https://doi.org/10.1016/j.cherd.2012.11.002
11. Zheng, D., He, X. and Che, D., "Cfd simulations of hydrodynamic characteristics in a gas–liquid vertical upward slug flow", ***International Journal of Heat and Mass Transfer***, Vol. 50, No. 21-22, (2007), 4151-4165. https://doi.org/10.1016/j.ijheatmasstransfer.2007.02.041
12. Abdulkadir, M., Hernandez-Perez, V., Lo, S., Lowndes, I. and Azzopardi, B.J., "Comparison of experimental and computational fluid dynamics (CFD) studies of slug flow in a vertical riser", ***Experimental Thermal and Fluid Science***, Vol. 68, (2015), 468-483. https://doi.org/10.1016/j.expthermflusci.2015.06.004
13. Adegoke, A.S. and Oyediran, A.A., "The analysis of nonlinear vibrations of top-tensioned cantilever pipes conveying pressurized steady two-phase flow under thermal loading", ***Mathematical and Computational Applications***, Vol. 22, No. 4, (2017), 44. https://doi.org/10.3390/mca22040044
14. Wang, L., Yang, Y., Li, Y. and Wang, Y., "Dynamic behaviours of horizontal gas-liquid pipes subjected to hydrodynamic slug flow: Modelling and experiments", ***International Journal of Pressure Vessels and Piping***, Vol. 161, (2018), 50-57. dhttps://doi.org/10.1016/j.ijpvp.2018.02.005
15. Mimouni, S., Fleau, S. and Vincent, S., "Cfd calculations of flow pattern maps and les of multiphase flows", ***Nuclear Engineering***

and Design, Vol. 321, (2017), 118-131. https://doi.org/10.1016/j.nucengdes.2016.12.009

16. Zhou, H., Guo, L., Yan, H. and Kuang, S., "Investigation and prediction of severe slugging frequency in pipeline-riser systems", ***Chemical Engineering Science***, Vol. 184, (2018), 72-84. https://doi.org/10.1016/j.ces.2018.03.050
17. Tay, B.L. and Thorpe, R.B., "Hydrodynamic forces acting on pipe bends in gas–liquid slug flow", ***Chemical Engineering Research and Design***, Vol. 92, No. 5, (2014), 812-825. https://doi.org/10.1016/j.cherd.2013.08.012
18. Montgomery, J., "Severe slugging and unstable flows in an s-shaped riser", (2002).
19. Malekzadeh, R., Henkes, R. and Mudde, R., "Severe slugging in a long pipeline–riser system: Experiments and predictions", ***International Journal of Multiphase Flow***, Vol. 46, (2012), 9-21.
20. Xie, C., Guo, L., Li, W., Zhou, H. and Zou, S., "The influence of backpressure on severe slugging in multiphase flow pipeline-riser systems", ***Chemical Engineering Science***, Vol. 163, (2017), 68-82. https://doi.org/10.1016/j.ces.2017.01.028
21. Assari, M., Basirat Tabrizi, H., Jafar Gholi Beik, A. and Shamesri, K., "Numerical study of water-air ejector using mixture and two-phase models", ***International Journal of Engineering, Transactions B: Applications***, Vol. 35, No. 2, (2022), 307-318. doi: 10.5829/ije.2022.35.02b.06.
22. Han, B., Ge, B., Wang, F., Gao, Q., Li, Z. and Fang, L., "Void fraction detection technology of gas-liquid two-phase bubbly flow based on convolutional neural network", ***Experimental Thermal and Fluid Science***, Vol. 142, (2023), 110804. https://doi.org/10.1016/j.expthermflusci.2022.110804
23. Graf, U. and Papadimitriou, P., "Simulation of two-phase flows in vertical tubes with the ctfd code flubox", ***Nuclear Engineering and Design***, Vol. 237, No. 20-21, (2007), 2120-2125. https://doi.org/10.1016/j.nucengdes.2007.02.011
24. Oshinowo, O., "Two-phase flow in a vertical tube coil", University of Toronto, (1971),
25. Jiang, Y. and Rezkallah, K.S., "A study on void fraction in vertical co-current upward and downward two-phase gas-liquid flow—i: Experimental results", ***Chemical Engineering Communications***, Vol. 126, No. 1, (1993), 221-243. https://doi.org/10.1080/00986449308936220
26. Usui, K. and Sato, K., "Vertically downward two-phase flow,(i) void distribution and average void fraction", ***Journal of Nuclear Science and Technology***, Vol. 26, No. 7, (1989), 670-680. https://doi.org/10.1080/18811248.1989.9734366
27. Nguyen, V.T., "Two-phase, gas-liquid concurrent flow: An investigation of holdup, pressure drop and flow pattern in a pipe at various inclinations", ResearchSpace@ Auckland, (1975),
28. Vatani, M. and Domiri-Ganji, D., "Experimental examination of gas-liquid two-phase flow patterns in an inclined rectangular channel with 90 bend for various vertical lengths", ***International Journal of Engineering, Transactions A: Basics***, Vol. 35, No. 4, (2022), 685-691. doi: 10.5829/IJE.2022.35.04A.07.

Persian Abstract

چکیده

در این مقاله، شبیه‌سازی سه‌بعدی همراه با مشاهدات تجربی برای مطالعه جریان دو فازی در یک لوله عمودی انجام گرفت. برای تجزیه و تحلیل هوا و آب از نرم افزار Open FOAMاستفاده شد. جریان اصلی رو به پایین که در داخل یک لوله عمودی به قطر ۱۰ میلی متر بود در نظر گرفته شد. مطالعه شامل دو ورودی برای جریان بود: ورودی بالایی برای آب و ورودی جانبی برای هوا. چندین حالت با شار جرمی مختلف برای آب و هوا مورد مطالعه قرار گرفت. بر اساس فیزیک موضوع، شبیه سازی عددی وابسته به زمان در نظر گرفته شد. نتایج به‌دست‌آمده نشان داد که وقتی سرعت هوا دارای مقادیر کمتری است ، تکانه هوا نمی‌تواند بر تکانه آب که منجر به اسلاگ های کوچک می شود غلبه کند. هنگامی که سرعت جریان هوا بیشتر از آب بود، بر جریان آب تسلط داشت و در نتیجه می‌توانست جهت جریان اصلی را تحت تأثیر قرار دهد. همچنین نمودارهای سرعت روی خط مرکزی نشان می‌دهند که هر چه زمان پیش می رود، بزرگی سرعت مقدار قابل توجهی از نوسانات را تجربه می‌کند و نوسانات بزرگ در نزدیک خروجی رخ می‌دهد. مقایسه نتایج تجربی و عددی، با تقریباً ۹ درصد اختلاف، تطابق قابل قبولی را نشان داد. نتایج نشان داد که در مراحل اولیه، تخلخل با افزایش مقدار قابل توجهی همراه است. شدت این تغییر در مقادیر تخلخل در سرعت کمتر آب بیشتر بود. در واقع، با افزایش سرعت آب، نیروهای اینرسی مرتبط با فاز مایع نقش غالبی در هیدرودینامیک کلی جریان گاز- مایع پیدا می‌کنند. همچنین بدیهی است که نحوه جریان در موارد ۱، ۲ و ۳ مشابه است اما پس از مورد ٤، الگوی جریان متفاوت است. این تغییرات در موارد ٥ و ٦ بیشتر قابل توجه است.