J Ther Eng, Vol. 7, Supp 14, pp. 2107–2114, December, 2021

Journal of Thermal Engineering
Web page info: https://jten.yildiz.edu.tr
DOI: /10.18186/thermal.1051642

Journal of Thermal Engineering

Technical Note

Hydrodynamics of single bubble rising through water column using volume of fluid (VOF) method

Emad Qasem HUSSEIN[1], Farhan LAFTA RASHID[1], Ahmed KADHIM HUSSEIN[2], Obai YOUNIS[3,4,*]

[1]Mechanical Engineering Department, University of Kerbala, Kerbala, IRAQ.
[2]College of Engineering, Mechanical Engineering Department, University of Babylon, Babylon City, IRAQ.
[3]Department of Mechanical Engineering, College of Engineering at Wadi Addwaser, Prince Sattam Bin Abdulaziz University, KSA.
[4]Department of Mechanical Engineering, Faculty of Engineering, University of Khartoum, SUDAN.

ARTICLE INFO

Article history
Received: 14 March 2021
Accepted: 28 June 2021

Key words:
Hydrodynamics; Rising velocity; Single bubble; Dynamic mesh; VOF

ABSTRACT

The importance of air bubble dynamics in liquid is in some phenomena like chemical and biochemical processes in refinery units. The 2D Volume of Fluid (VOF) method together with the CFD technique were employed for simulating. The dynamic meshing technique is used to simulate the hydrodynamics of rising air bubble in liquid water column via the User Defined Function (UDF) code in the C++ environment was developed to evaluate bubble rising through the water column. The rising of air bubble through a stagnant water column has been considered and the influence of column dimension, bubble size, and aspect ratio on the rising velocity characterized is investigated. The obtained results showed that the bubble rising velocity increase with the bubble size and its shapes was transformed from ellipsoidal-to-ellipsoidal cap shape. The rising velocity of air bubbles was affected by the column diameter. It was observed that the air bubble moving toward the top of the water column with oscillation for all cases. A good agreement was obtained between the rising velocity predicted in the simulation with that obtained from the literature.

Cite this article as: Hussein EQ, Rashid FL, Hussein AK, Younis O. Hydrodynamics of single bubble rising through water column using volume of fluid (VOF) method. J Ther Eng 2021;7(Supp 14):2107–2114.

INTRODUCTION

The phenomenon of air-water two-phase flow appeared in many applications such as in petroleum, chemical, nuclear power plant, solar energy, and biogas energy [1–4]. The motion of air bubbles in a liquid column can be considered very complex because of the high viscosity and density ratios. In addition, it was complicated to formulate an

***Corresponding author.**
*E-mail address: o.elamin@psau.edu.sa, oubeytaha@hotmail.com

This paper was recommended for publication in revised form by Regional Editor Ahmet Selim Dalkilic

Published by Yıldız Technical University Press, İstanbul, Turkey
Copyright 2021, Yıldız Technical University. This is an open access article under the CC BY-NC license (http://creativecommons.org/licenses/by-nc/4.0/).

J Ther Eng, Vol. 7, Supp 14, pp. 2107–2114, December, 2021

all right mathematical model to be used in calculating the rising velocity of air bubble in different system parameters and physical properties. The behavior of a rising air bubble is a function of bubble size and air-water properties such as density, surface tension, and viscosity [5]. The rising of air bubble can be presented by experimental investigations, but there is main difficulty in the quantitative estimation that the multiphase flow studies demonstrate complex problems [6]. Many numerical techniques and computing power were advanced like computational fluid dynamics (CFD) which became a good approach of studying the behavior of rising air bubble in vertical water column [7–10]. The method of the volume of fluid (VOF) was an effective way for simulation of the two-phase flow [11]. The volume of the fluid method was utilized for interfacial tracking motion in all phases, which is included in the continuum surface force (CSE) equation and governing equation at the interface [12–17]. Therefore, the method of level-set (LS) is coupled with the method of the volume of fluid to enhance the interfacial pursuit and bubble shape [18–20]. Yong et al. [20–23] used the volume of fluid method to simulate the distortion of a bubble rising under very high pressure. Hua et al. [24] used a direct numerical simulation of three dimensions for simulating the interaction of two bubbles in a stagnant viscous fluid. Chen et al. [25] used the method of the volume of fluid to simulate the rising of a single bubble in a stagnant water column and the interaction of the gas-water interface. Huang et al. [26] used the method of the volume of fluid with a momentum equation to simulate the shape geometry and rising manner of the twin bubbles of horizontal arrangement in a steady liquid. Pär et al. [27] revealed a one-dimensional model of marine two-phase gas and the interaction between the dissolved and free gas phases and the gas liberated to the atmosphere. Kishor et al. [28] presented a numerical simulation of bubble hydrodynamic and pressure drop due to friction associated with gas-liquid two-phase flow in T-junction microchannel. Zhang et al. [29] experimentally investigated the transient motion of a single bubble in a highly viscous fluid, they successfully provided empirical correlations for total drag coefficient and joint unsteady force coefficient which are in very good agreement with the experimental results.

The objective of this research is to implement the theoretical investigation of different air bubble sizes raised in stagnant water contained by different column shapes. Results from this study could be useful to understand the true physical phenomena and design of a water column. Finally, it proposes innovative criteria for the selection of the multiphase flow model on CFD simulation.

MATHEMATICAL MODEL

Case Study

In this work, the 2D domain is employed to explore the behavior of rising single air bubble in the stagnant water column. The dimensions of the column are: height of (100–150) mm and width of (20–40) mm as presented in Figure (1). The single air bubble is decreed at the center and 10 mm height from the base of the column at the first stage of simulation. In this work, four bubble diameters of 5, 6, 7, and 8 mm are studied. The air bubble in the compliant water will raises under the buoyancy force action [30].

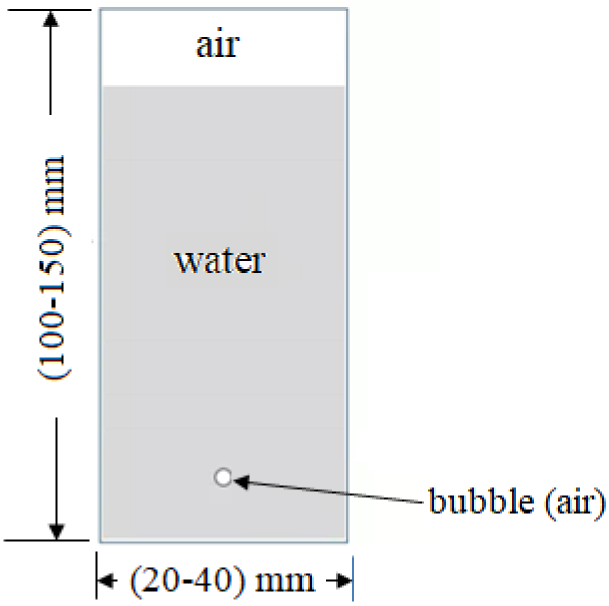

Figure 1. Schematic of possible flow regimes in bubble columns.

In this work, the primary phase is water (liquid) and the secondary phase is air (gas). The main *physical properties are used to observe* and describe the matter of the single air and water are tabulated in Table (1) [31].

Governing Equations and Modeling

In this paper, a comprehensive CFD modeling has been employed to cover all details of two-phase flow during the operation of a moreover, a user-defined function has been used to complete the Fluent code to simulate the phase change material.

Continuity Equation

The unsteady state continuity equation of fluid is based upon the principle of mass conservation. The

Table 1. Physical properties of test fluids at atmospheric conditions [31]

Fluid	Air	Water
Density, kg/m^3	1.22	999
Viscosity, *Pa.s*	0.018	0.89
Surface tension, *mN/m*	–	72.0

general formula of the unsteady-state continuity equation is [31, 32]:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \bar{v}) = 0 \quad (1)$$

Equation of Gas Volume Fraction

The unsteady state gas volumetric fraction equation in 2D can be presented by the following equation [30]:

$$\frac{\partial \alpha_g}{\partial t} + \frac{\partial (u_{g,x}\alpha_g)}{\partial x} + \frac{\partial (u_{g,y}\alpha_g)}{\partial y} = 0 \quad (2)$$

The equation of liquid water volume fraction in the similar formula of gas (air) volume fraction, and the sum of two is equal to unity:

$$\alpha_1 + \alpha_g = 1 \quad (3)$$

Where α_1 and α_g, are volumetric fraction of liquid and fluid phase respectively, x and y, are the horizontal and vertical axis, and t is the time. Solution of the above equation for the volume fraction of one of the phases is used to track the interface between the phases.

Momentum Equation

In Newton's second law, it was stated that the sum of the forces which are acting on an element of fluid is equal to its acceleration rate of momentum change. This relates to the momentum equation, the general form of the momentum equation [31]:

$$p\left[\frac{\partial \bar{u}}{\partial t} + (\bar{u}.\nabla)\bar{u}\right] = -\nabla p + \nabla.(2\mu D) + \rho \vec{g} + \vec{F} \quad (4)$$

Where g, p, μ, and u are the acceleration of gravity, pressure, dynamic viscosity, and velocity, respectively, and D is the stress tensor which can be presented as:

$$D_{ij} = \frac{1}{2}\left[\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right] \quad (5)$$

Operating Conditions

Both the base and sidewalls of the region are assumed as a non-slip boundary condition and the top wall as pressure outlet boundary condition. The operating pressure is set to be equal to ambient pressure. The ambient pressure, i.e. 101325 pa is set as the operating pressure, the acceleration of gravity is assigned along the y-direction. ANSYS Fluent commercial package was utilized for solving the continuity, momentum, and fractions of the volume of fluid [32, 33]. The flow equations were of the second-order upwind scheme. The pressure velocity coupling was solved by splitting operations algorithm [34]. The solution stability will attain a convergence without a significant loss [35]. The body force weight sketch and the implicit force curing were used to solve the pressure and improve the solution convergence. A grid independence study is carried out to find the grid size required for simulating an initially static spherical bubble of initial diameter is considered for the study. Moreover, a user-defined function has been used to complete the fluent code to simulate the phase change material. A time step of 10^{-4} was assigned to simulate the transient flow based on the explicit scheme.

RESULTS AND DISCUSSION

The air bubble rising through the water column has been considered and the impact of column dimension, bubble size, and aspect ratio on the rising velocity and shapes of the air bubble is investigated. The case research of single air (gas) bubble rising in viscous water (liquid) was investigated in various applications of multiphase flow. The obtained CFD results were compared against the experimental results data of 4mm bubble in low Morton numbers for purpose of validation as shown in Figure (2). The results of this work show that CFD is a necessary tool for the evaluation and agree well with that obtained from the experimental results of Zhang et al. [29].

The CFD results of a single air bubble rising through a water column with different bubble sizes of (dB = 5, 6, 7, and 8 mm) are depicted in Figure (3).

The rising velocity of a single air bubble increases with increasing the size of the bubble due to the impact of the increased buoyancy force. When the bubble is wobbling, the rising velocity at the beginning decreases and after that increases smoothly. During the air bubble oscillation, the rising bubble velocity was decreased at the zone of the oscillating spherical cap. At the end, the rising bubble velocity was increased at the zone of the spherical cap due to the

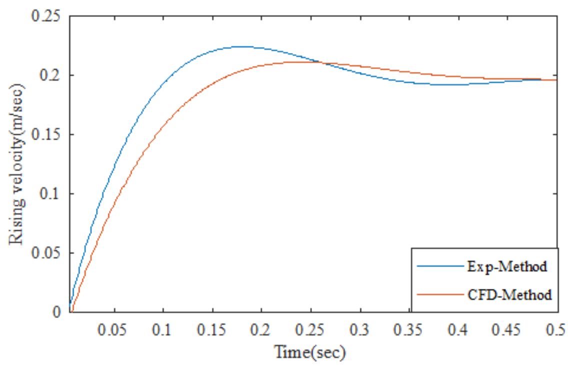

Figure 2. Experimental and numerical rising velocity varying with time.

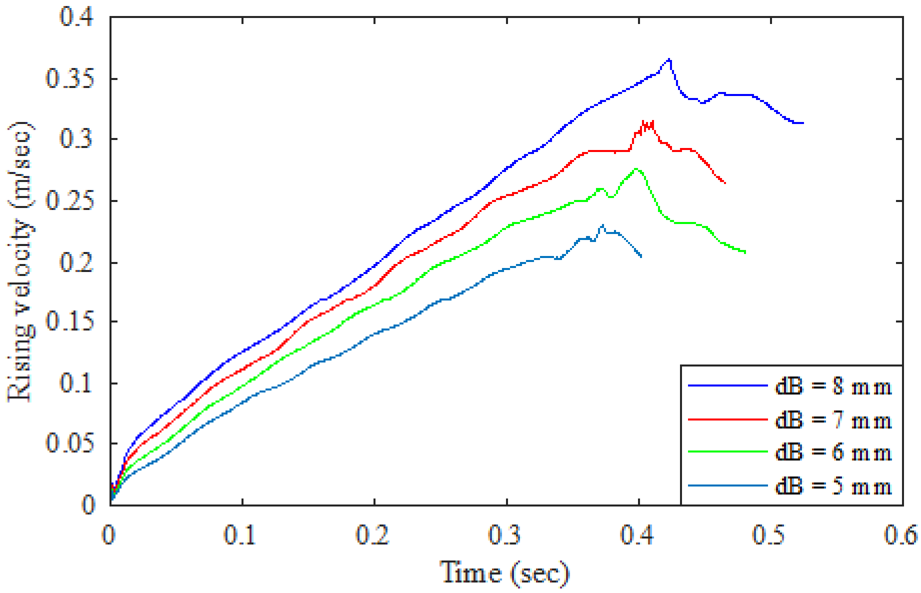

Figure 3. Time evolution of rising velocity at different bubble size.

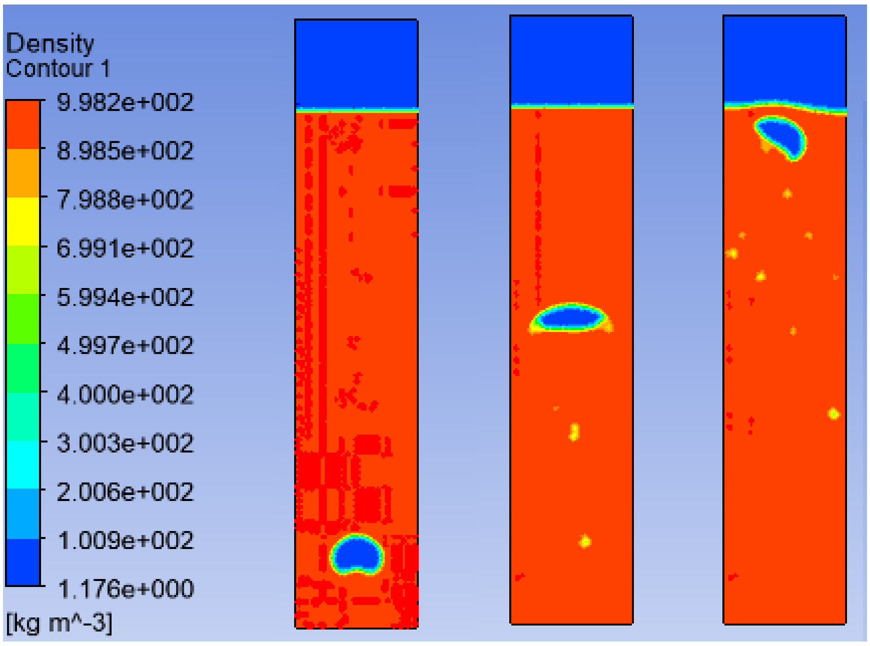

Figure 4. Density contour for large bubble size with different time.

reduction in the shape oscillation and reached a constant value.

For large bubbles, their velocity tends to increase but they are not stable and tend to subdivide into smaller bubbles as shown in Figure (4). The external field stresses deform the bubble surface which leads to zigzag motion by the asymmetry of the pressure field in the periphery of the bubble. During the zigzag movement, the bubble reverses its axis or form oscillation until reaching the top of the domain, as shown in Figure (5). For the latter case, the bubble rises faster with shape oscillation due to a vehement vortex of liquid jet acts succeed the bubble.

The result concludes that the bubble shapes vary with the diameter and this is caused by the varying drag forces. Therefore, the bubble rising velocity increases with the size of the bubble, and the bubble shape geometry change from

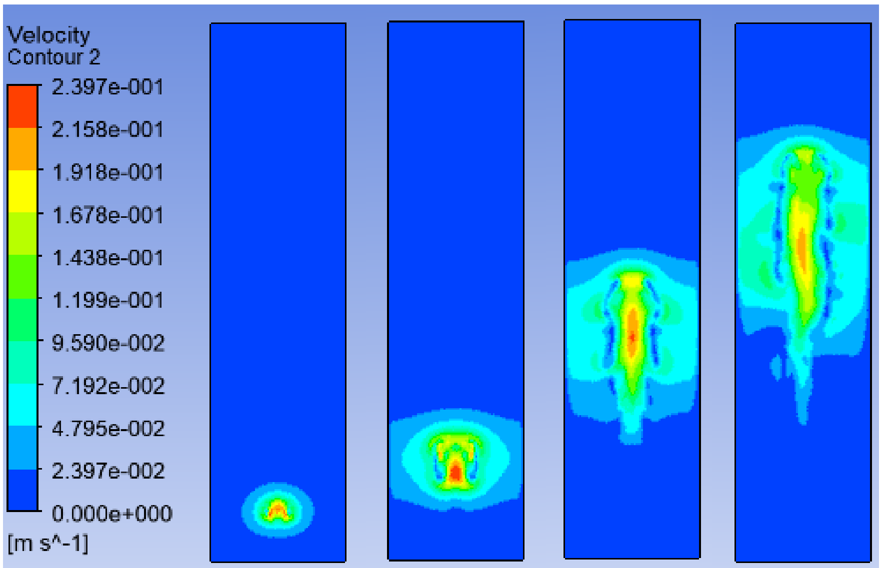

Figure 5. Rising velocity contour with different time of simulation.

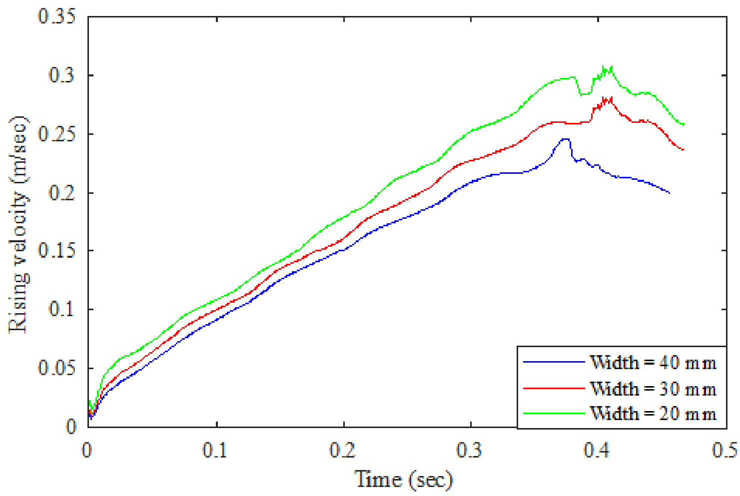

Figure 6. Rising velocity with time at different tube size.

an ellipsoidal form to an ellipsoidal cap. The effect of the width column on the bubble characteristic is an investigation for three different domains at a fixed height of 100mm and varying width (20,30 and 40mm) for a fixed bubble diameter of 5mm. The single bubble velocity varied with time as presented in Figure (6). The rising velocity of the air bubble was influenced by the diameter of the water column. however, this influence was diminished when the ratio of bubble diameter to the water column diameter is smaller. The rising velocity was affected by the water column diameter when increasing the ratio of bubble diameter to the water column diameter, due to the wall column effect.

The variation of bubble rise velocity with column height (100–150) mm for a fixed width of 30mmof the domain is given in Figure (7) for a fixed bubble diameter of 5mm. The domain height is sufficient for bubble dynamic simulation in all cases of study, and the rising velocity decreases with increase column height due to the increase in bubble residence time at the top of the water column, which leads to enhance the mean bubble diameter to bubble coalescence.

The initial shape of air bubble in liquid water with a different aspect ratio of (0.75,1 and 1.25), as shown in Figure (8).

Figure (9) reveals bubble rising velocity variation with time at different bubble aspect ratios. It is clear that the higher aspect ratio gives higher rising velocity at the initial time due to the low flow drag. The development of the final bubble shape is affected by the initial bubble shape in

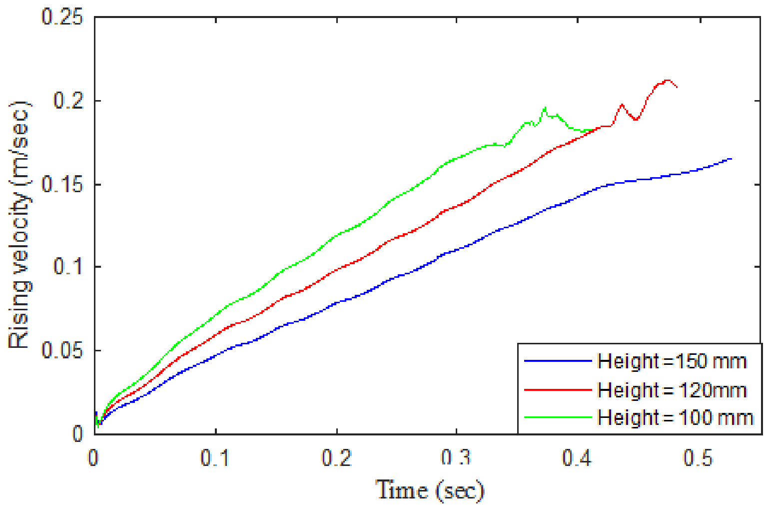

Figure 7. The rising velocity with time at different column height.

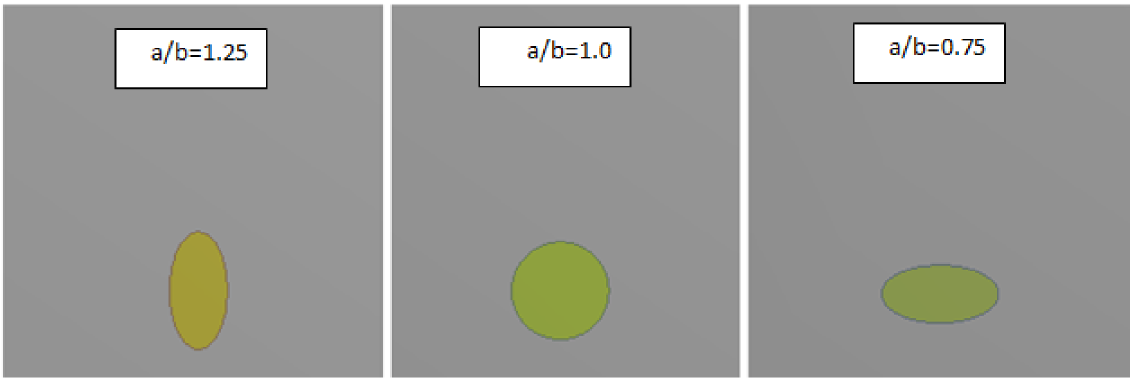

Figure 8. Bubble dimension with the different aspect ratio.

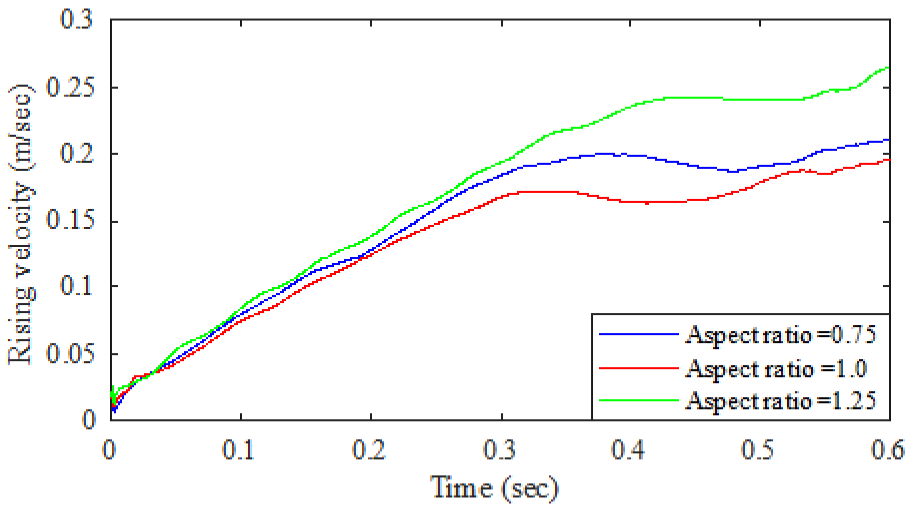

Figure 9. The rising velocity with time at a different aspect ratio.

the flow regime. When the aspect ratio of the initial bubble shape is lower, the cap bubble is transformed into a spherical cap bubble, and when the aspect ratio of the initial bubble shape is higher, the toroidal bubble is developed. The air bubble rising velocity is affected by the initial bubble shape. The results concluded from this study could be useful to understand the true physical phenomena and design of an air bubble water column.

CONCLUSION

In this work, four different sizes (diameters) of air bubbles (5, 6, 7, and 8 mm) in the water column were numerically studied by using the VOF method. The following points were concluded from the present work:

1. The air bubble rising velocity of the present CFD models agreed well with the experimental results of Chakraborty et al. [35]. The bubble rising velocity increased with increasing the bubble size, and it was influenced by the water column shape geometry.
2. The oscillations were found to be more pronounced for lower bubble size and the amplitude is observed to decrease with increasing the diameter of the air bubble.
3. The maximum deviation in rising velocities in all cases of study decreased with the increase in bubble diameter.
4. The velocity of bubble rising is affected by the initial shape of the bubble, it decreases with increase column height due to the increase in bubble residence time at the top of the water column.
5. The bubble shapes vary with the diameter due to the effect of varying drag forces.

ACKNOWLEDGMENTS

This publication was supported by the Deanship of Scientific Research at Prince Sattam bin Abdulaziz University, Alkharj, Saudi Arabia.

AUTHORSHIP CONTRIBUTIONS

Authors equally contributed to this work.

DATA AVAILABILITY STATEMENT

The authors confirm that the data that supports the findings of this study are available within the article. Raw data that support the finding of this study are available from the corresponding author, upon reasonable request.

CONFLICT OF INTEREST

The author declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

ETHICS

There are no ethical issues with the publication of this manuscript.

REFERENCES

[1] Xie Y B, Liu D P, Yang L, Yang M. Experimental study on the growth characteristics of natural gas hydrateformation on suspended gas bubbles in distilled water and tap water. Chem Ind Eng Prog 2017;36:129–135.
[2] Yang H, Zhu C Y, Ma Y G, Gao X Q. Continuous formation and coalescence of bubbles through double-nozzle in non-newtonian fluid. Chem Eng 2016;44:37–41.
[3] Painmanakul P, Loubiere K, Hebrard G, Buère P. Study of different membrane spargers used in waste water treatment: characterisation and performance. Chem Eng Process 2004;43:1347–1359. [CrossRef]
[4] Gnyloskurenko S V, Byakova A V, Raychenko O I, Nakamura T. Influence of wetting conditions on bubble formation at orifice in an inviscidliquid. Physicochem Eng Asp 2003;218:73–87. [CrossRef]
[5] Kulkarni A A, Joshi JB. Bubble formation and bubble rise velocity in gas-liquid systems: a review. Ind Eng Chem Res. 2005;44:5873–5931. [CrossRef]
[6] Dong Z, Li W, Song Y A. Numerical investigation of bubble growth on and departure from a superheated wall by lattice boltzmann method. Int J Heat Mass Transf 2010;53:4908–4916. [CrossRef]
[7] Zhang ZY, Jin LA, He SY, Yuan ZJ. Study on Coupling Models for Bubble Floatation Accompanied with heat and mass transfer. J Chem Eng Chin Univ 2018;32:358–367.
[8] Yoo D H, Tsuge H, Terasaka K, Mizutani K. Behavior of bubble formation in suspended solution for an elevated pressure system. Chem Eng Sci 1997;52:3701–3707. [CrossRef]
[9] Ding Y D, Liao Q, Zhu X, Wang H, Liu Z H. Characteristics of bubble growth and water invasion at the permeable sidewall in rectangular liquid flow channel. J Mech Eng 2013;49:119–124. [CrossRef]
[10] Du YH, Xiong KW, Zhang Y, Nie XT, Zhou M, et al. Analysis of numerical simulation on horizontal arrangement equal bubbles rising. Chin J Comput Mech 2016;33:889–894.
[11] Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. J Comput Phys 1981;39:201–225. [CrossRef]
[12] Rabha SS, Buwa VV. Volume-of-fluid (VOF) simulations of rise of single/multiple bubbles in sheared liquids. Chem Eng Sci 2010;65:527–537. [CrossRef]
[13] Chakraborty I, Biswas G, Ghoshdastidar P. A coupled level-set and volume-of-fluid method for the

buoyant rise of gas bubbles in liquids. Int J Heat Mass Transf 2013;58:240–259. [CrossRef]
[14] Szewc K, Pozorski J, Minier J P. Simulations of single bubbles rising through viscous liquids using smoothed particle hydrodynamics. Int J Multiph Flow 2013; 50: 98–105. [CrossRef]
[15] Ma D, Liu M, Zu Y, Tang C. Two-dimensional volume of fluid simulation studies on single bubble formation and dynamics in bubble columns. Chem Eng Sci 2012;72:61–77. [CrossRef]
[16] Buetehorn S, Volmering D, Vossenkaul K, Wintgens T, Wessling M, Melin T. CFD Simulation of single-and multi-phase flows through submerged membrane units with irregular fiber arrangement. J Membr Sci 2011;384:184–197. [CrossRef]
[17] Yang N, Wu Z, Chen J, Wang Y, Li J. Multi-scale analysis of gas-liquid interaction and cfd simulation of gas-liquid flow in bubble columns. Chem Eng Sci 2011;66:3212–3222. [CrossRef]
[18] Gerlach D, Alleborn N, Buwa V, Durst F. Numerical simulation of periodic bubble formation at a submerged orifice with constant gas flow rate. Chem Eng Sci 2007;62:2109–2125. [CrossRef]
[19] Yan K, Che D. A coupled model for simulation of the gas-liquid two-phase flow with complex flow patterns. Int J Multiph Flow 2007;36:333–348. [CrossRef]
[20] Li Y, Zhang JP, Fan LS. Discrete-phase simulation of single bubble rise behavior at elevated pressures in a bubble column. Chem Eng Sci 2000;55:4597–4609. [CrossRef]
[21] Li Y, Zhang JP, Fan LS. Numerical studies of bubble dynamics in gas-liquid-solid fluidization at high Pressures. Powder Technol 2001;116:246–260. [CrossRef]
[22] Yang G Q, Du B, Fan L S. Bubble formation and dynamics in gas-liquid-solid fluidization—a review. Chem Eng Sci 2007;62:2–27. [CrossRef]
[23] Zhang J P, Li Y, Fan LS. Numerical studies of bubble and particle dynamics in a three-phase fluidized bed at elevated pressures. Powder Technol 2000;112:46–56. [CrossRef]
[24] Hua JS, Lin P, Jan FS. Numerical simulation of gas bubbles rising in viscous liquids at high reynolds number. Contemp Math 2008;466:1–18. [CrossRef]
[25] Chen L, Li Y. A numerical method for two-phase flows with an interface. Environ Model Softw 1998;13:247–255. [CrossRef]
[26] Huang Y, Gao P, Wang C. Experimental and numerical investigation of bubble–bubble interactions during the process of free ascension. Energies 2019;12:1977. [CrossRef]
[27] Pär J, Bénédicte F, Anna S, Knut O D, Anders O. A new numerical model for understanding free and dissolved gas progression toward the atmosphere in aquatic methane seepagesystems. Limnol Oceanogr Methods 2019;17:223–239. [CrossRef]
[28] Kishor K, Chandra AK, Khan W, Mishra PK, Siraj AM. Numerical study on bubble dynamics and two-phase frictional pressure drop of slug flow regime in adiabatic t-junction square microchannel. Chem Biochem Eng Q 2017;31:275–291. [CrossRef]
[29] Zhang L, Yang C, Mao ZS. Unsteady motion of a single bubble in highly viscous liquid and empirical correlation of drag coefficient. Chem Eng Sci 2008;63:2099–2106. [CrossRef]
[30] Liang-Shih F, Katsumi T. Bubble Wake Dynamics in Liquids and Liquid-Solid Suspensions. Ohio State University: Butterworth-Heinemann; 1990.
[31] Webber N B. Fluid Mechanics for Civil Engineers. London: Chapman and Hall; 1971.
[32] A. Fluent, "User's Guide", ANSYS 14.0, 2010. Available from: http://www.pmt.usp.br/academic/martoran/notasmodelosgrad/ANSYS%20Fluent%20Users%20Guide.pdf.
[33] ANSYS Inc. Fluent, 2010. Available from: http://www.Ansys.Com/Products/Fluid-Dynamics/Fluent/
[34] Akhtar A. CFD Simulations for Continuous Flow of Bubbles through Gas-Liquid Columns: application of VOF method. Chem Prod Process Model 2007;2:1–19. [CrossRef]
[35] Issa RI. Solution of the implicitly discretized fluid flow equations by operator-splitting. J Comput Phys 1986;62:40–65. [CrossRef]