FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
May 31, 2025

Study of Rise of Multiple Bubbles in Quiescent Liquid using OpenFOAM Software

Hyzam Kaliyadan
B.Tech Mechanical Engineering, School Of Engineering, CUSAT
Faculty Guide: Prof. Yogesh Ahire
Sanjivani College of Engineering, Kopargaon
FOSSEE Mentor: Mr. Nikhil Chitnavis

Synopsis

The numerical study of multiple bubbles rising in quiescent liquids reveals intricate dynamics influenced by various factors such as bubble size, spacing, and liquid velocity etc. The volume of fluid (VOF) method is commonly employed to simulate these interactions, providing insights into coalescence, breakup, repulsion, the resulting flow structures, and the effects of liquid properties on bubble behaviour. The reference paper used for this migration is: Wassim Abbassi, Sonia Besbes, Habib Ben Aissia, Jean Yves Champagne, Study of the rise of a single/multiple bubbles in quiescent liquids using the VOF method, Journal of the Brazilian Society of Mechanical Sciences and Engineering (2019) 41:272. https://doi.org/10.1007/s40430-019-1759-y

1 Introduction

The paper referred for this research migration project examines the dynamics of a single and multiple bubbles in quiescent liquids using the Volume of Fluid Method. The authors investigate the effects of liquid viscosity, bubble size and initial bubble configuration on the bubble dynamics and interactions. It is found that the bubble rising trajectory changes from rectilinear to a zig-zag trajectory when the liquid viscosity decreases. When three bubbles rise side by side, the magnitude of bubble shape oscillations increases compared to a single bubble. For two in-line bubbles with different sizes, the larger bubble was observed to always have a strong effect on the smaller one for each initial configuration. Ansys Fluent was used for the numerical simulations in the reference paper. This research migration project intends to reproduce the results obtained in the reference paper using the interFoam solver in OpenFOAM.

2 Governing Equations and Models

The interFoam solver, which employs the Volume of Fluid (VOF) method to capture the interface between two immiscible fluids, has been used for this project. In this method, the fluid is identified by the volume fraction function α_k. Thus, for each phase k in a computational domain cell, the volume fraction α is tracked throughout the domain, which means the fraction of the volume filled by the fluid in the grid to achieve the goal. For gas-liquid two-phase flow configuration, the volume fraction α is defined as follows:

$$\alpha = \frac{\text{Volume of fluid in unit}}{\text{Volume of unit}} \tag{1}$$

where

$$\alpha = \begin{cases} 0 & \text{In the bubble (gas)} \\ 0 < \alpha < 1 & \text{At the interface} \\ 1 & \text{In the liquid} \end{cases}$$

The solver solves the Navier Stokes equations for two incompressible, immiscible fluids where the material properties remain constant in the region filled by one of the two fluid except at the interphase. Here the simulations are carried out in a 2D domain. Using the VOF method, the behaviour of both phases can be represented by a single set of continuity and momentum equations which incorporates surface tensions effects. For an incompressible Newtonian Fluid, these equations are:

Continuity Equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2}$$

Momentum Equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j u_i\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{t_{ij}}\right) + \rho g_i + f_{\sigma i} \tag{3}$$

Here the simulation is laminar, so the turbulent term $\tau_{t_{ij}}$ is set to 0.

The density ρ is defined as follows:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{4}$$

The surface tension force, $f_{\sigma i}$, is modeled as a continuum surface force (CSF). It is calculated as follows:

$$f_{\sigma i} = \sigma\kappa\frac{\partial \alpha}{\partial x_i} \tag{5}$$

where $\kappa = -\nabla \cdot \mathbf{n}; \mathbf{n} = \left(\frac{\nabla\alpha}{|\nabla\alpha|}\right)$

In order to know where the interphase between the two fluids are, an additional equation for α has to be solved which is given by:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_j}(\alpha u_j) = 0 \tag{6}$$

3 Simulation Procedure

3.1 Geometry and Mesh

The computational domain was a quadrangle-structured 2D column of dimensions 100mm width × 200mm height, with the third dimension having one cell of thickness of 1mm. The patches are named as walls, top & frontAndBack. Since this is a 2D Domain, the frontAndBack patch type was set to empty. The grid size was chosen to be 0.2 mm^2. Thus there were 500 $\times$1000 = 1500000 cells in the domain. This domain was created using the blockMesh utility in OpenFOAM.

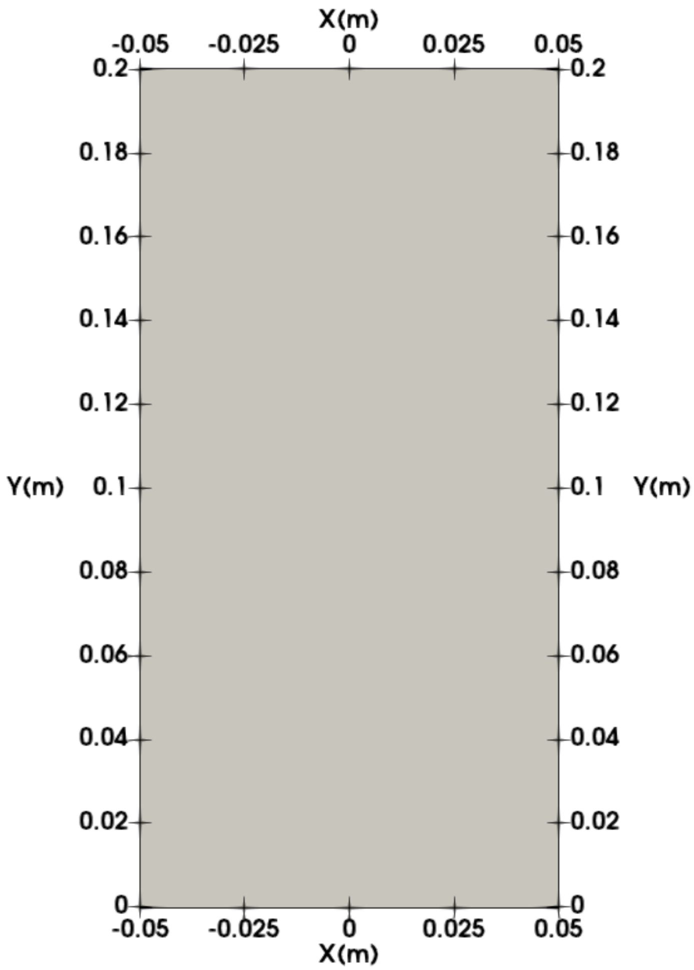


Figure 1: Schematic of the computational domain used for the bubble rise simulation.

3.2 Initial and Boundary Conditions

For understanding the effect of bubble size in its dynamics, bubbles of sizes 2, 4, and 6 mm were adopted. In all the simulations, the bubble(s) has been positioned in the domain using the setFields utility. The co-ordinates of bubble placement are as follows:

Table 1: Co-ordinates of Bubble centre

Configuration	Co-ordinates (mm)
Single Bubble	(0,5)
Two Bubbles in-line	(0,5) , (0,17)
Three Bubbles side-by-side	(0,5) , (-12,5) , (12,5)

The column is filled with the liquid upto 180mm and the top region is air. The phase fraction field alpha.phase1 was used to define the regions of air and liquid, with alpha.phase1 = 0 for air and alpha.phase1 = 1 for liquid.

Three different liquids were considered for the analysis, in order to understand the effect of parameters like liquid density, viscosity and surface tension on bubble dynamics. The physical properties of the working fluids are as follows:

Table 2: Physical properties of working liquids at 25°C

System No.	Solution	μ_L ($\times 10^3$ Pa·s)	Surface Tension ($\times 10^3$ N/m)	Density (kg/m^3)
S-1	Deionized water	0.95	72.0	997
S-2	64% glycerin in water	12.3	66.7	1162
S-3	72% glycerin in water	24.8	66.3	1184

The properties of air at 25°C are:

Table 3: Physical Properties of Air

Parameter	**Value**	**Unit**
Dynamic Viscosity (ν_{air})	1.562×10^{-5}	Pa·s
Density (ρ)	1.184	kg/m^3

The boundary conditions for these simulations were set as follows:

Table 4: Boundary conditions used in the simulation

Parameter	**Patch**	**Type**	**Condition**
alpha.phase1 (internalField - uniform 0)	walls	zeroGradient	
	top	inletOutlet	inletValue - uniform 0 value - uniform 0
p_rgh (internalField - uniform 0)	walls	fixedFluxPressure	uniform 0
	top	prghTotalPressure	value - uniform 0 p0 - uniform 0
U (internalField - uniform 0)	walls	noSlip	
	top	pressureInletOutletVelocity	uniform 0

In all the cases, the bubble was initialized with an initial velocity of 0.1 m/s. The effects of gravity is considered in this simulation. In order to take advantage of multiple processors, the case was decomposed using the decomposePar utility which partitions the computational domain into the specified number of sub-domains, and each sub-domain is allocated to a separate processor. This allows the simulation to be run in parallel, significantly reducing the overall computational time.

3.3 Solver

The solver interFoam was identified as the best suited solver for simulations. This is an incompressible, transient solver specfically designed for two-phase flows with a sharp interface between immiscible fluids, utilizing a Volume of Fluid (VOF) method. The solver implements a PISO (Pressure Implicit with Splitting of Operators) algorithm for arriving at the solution.

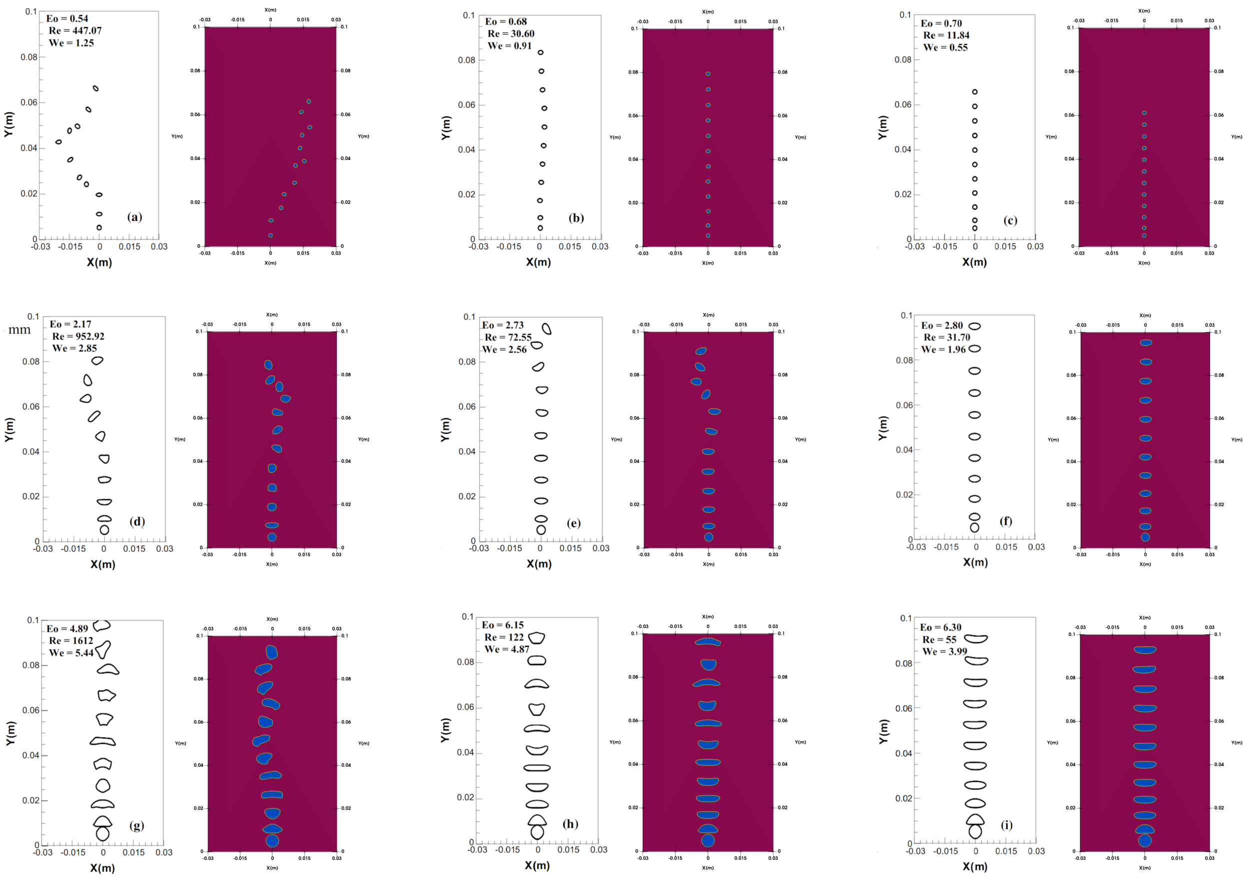
The Gauss QUICK Scheme was used for the discretization of the spatial derivatives in Eqns (3) and (6), while the temporal derivatives were discretized using Gauss linear scheme. In all simulations, the maximum Courant number was limited to 0.25.

4 Results and Discussions

4.1 Single Bubble

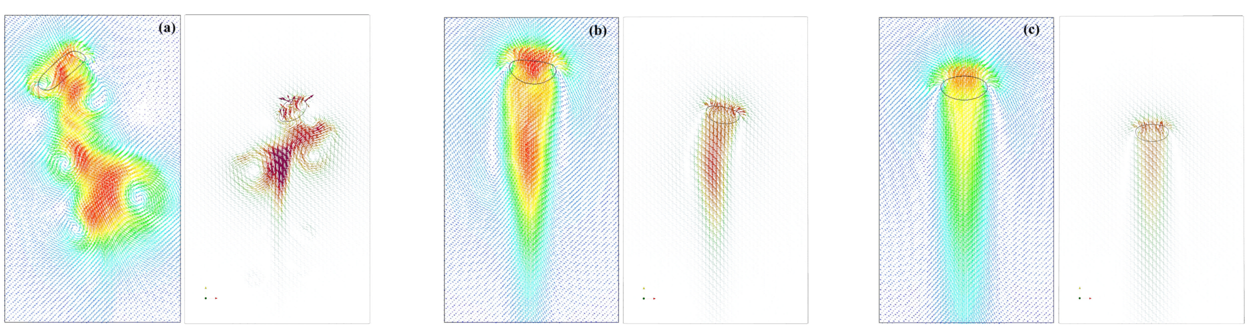
The rise of bubbles are governed by several forces such as surface tension forces, inertial forces, viscous drag forces and buoyancy forces. The timestep between two successive bubble images is Δt=0.05s. It can be observed that bubbles of smaller diameter have the least deformation, which is due to higher surface tension forces acting on the small bubble. The bubbles of smaller diameters deviates the most from the axis, because their lower inertia makes them more susceptible to the lateral forces generated from the unstable wake. Also it can be observed that as viscosity of liquid increases, the extent of deviation from the vertical axis decreases, due to the increase of viscous drag forces which dominates the inertial force.

When comparing the results from the original paper, it is observed that the bubble trajectory in Solution 1 deviates from those presented in the paper (fig. 2). This could be due to the differences in the numerical discretization schemes and solver settings. It could be observed that as liquid density increases, the bubble rise velocity decreases (fig. 3). Analysing the vorticity fields induced by the single bubbles in fig. 4, it can be observed that there is a stronger vorticity field present in bubble of solution 1, and its strength decreases as liquid density decreases.



Solution 1 Solution 2 Solution 3

Figure 2: Comparison of results of shapes and trajectories of bubbles of diameters 2,4 and 6mm in three different solutions, as obtained in the paper vs those conducted in OpenFoam.



Solution 1 Solution 2 Solution 3

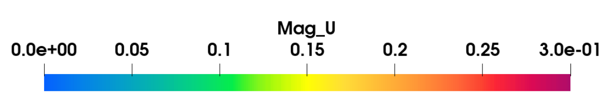


Figure 3: Comparison of instantaneous liquid velocity field induced by a single bubble rising in three different solutions, as obtained in the paper vs those conducted in OpenFoam.

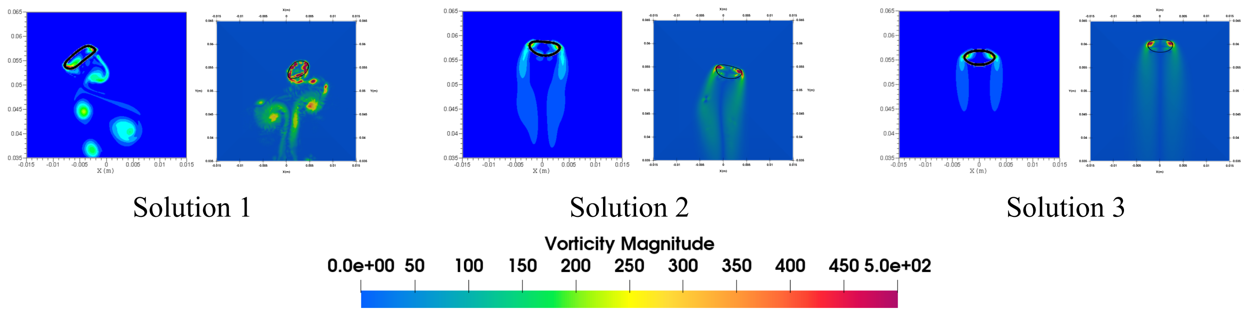
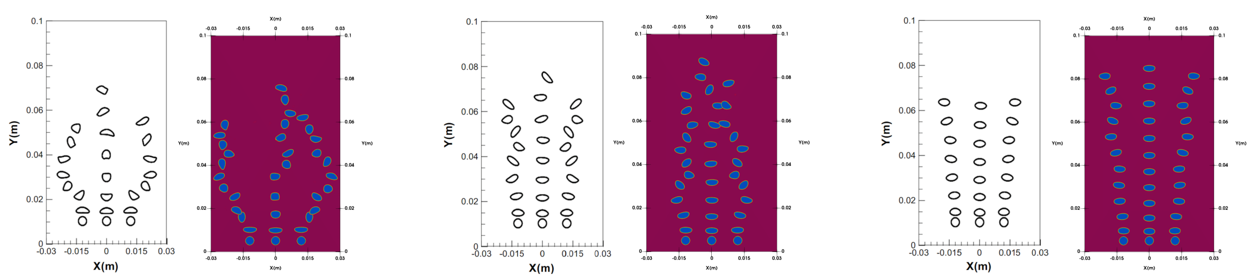


Figure 4: Comparison of results of instantaneous vorticity field induced by a single bubble in three different solutions, as obtained in the paper vs those conducted in OpenFoam

4.2 Three bubbles rising side by side

The fig. 5 depicts the difference in bubble rise trajectory as liquid density increasess, with the bubbles in solution 1 showing the maximum deviation from the vertical axis due to higher inertial force effect. In solution 3, the viscous drag force dominates, which prevents excessive zig-zag motion, nevertheless the bubbles in either ends of the domain divert away from the bubble in the middle, due to the effect of vortices generated behind the bubbles. The fig. 6 and fig. 7 depict the instantaneous velocity field and vorticity fields generated by the bubbles. In the OpenFoam simulaions, the general trend of decreasing velocity and decreasing vorticity as liquid density increases has been captured, with the velocity field data in good agreement with the data presented in the paper.



Solution 1 Solution 2 Solution 3

Figure 5: Comparison of trajectory and shape of three bubbles of diameter = 4mm rising side by side, as obtained in paper vs results obtained in OpenFoam.

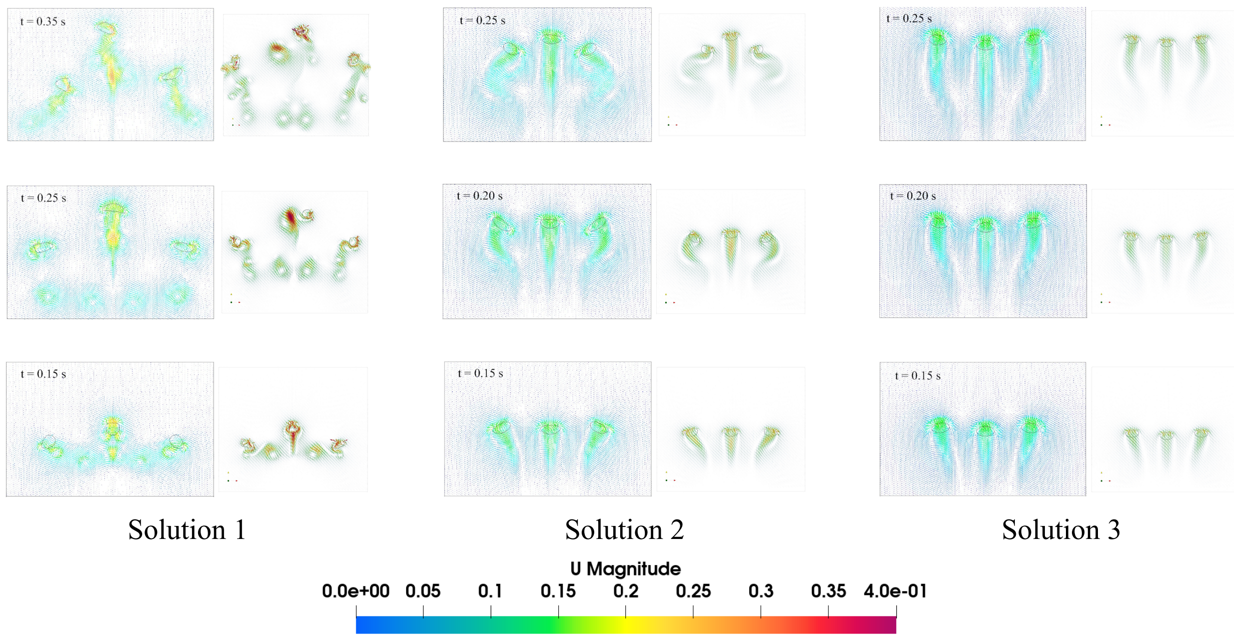


Figure 6: Comparison of instantaneous velocity field induced by three bubbles of diameter = 4mm rising side by side, as obtained in the paper vs results obtained in OpenFoam

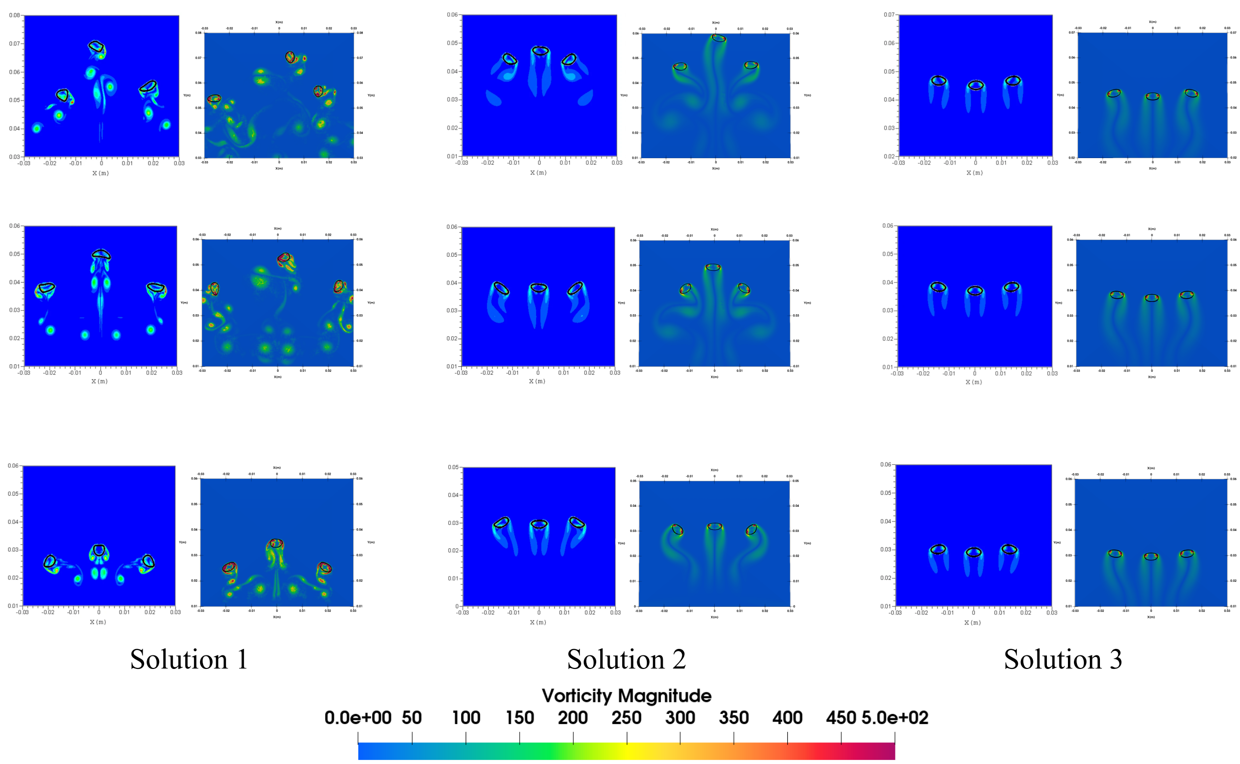


Figure 7: Comparison of results of instantaneous vorticity field induced by three bubbles rising side by side in three different solutions, as obtained in the paper vs those conducted in OpenFoam.

4.3 Two Bubbles rising in-line

4.3.1 Both Bubbles of Same Diameter

From the fig. 8, it can be observed that for two bubbles of the same diameter rising in-line, they initially rise independently of each other, but as the trailing bubble comes within the wake of the leading bubble, the wake reduces the viscous forces acting on the trailing bubble, which causes it to accelerate. This leads to eventual coalescence of the two bubbles. The instantaneous velocities of the OpenFoam simulation are in good agreement with the values obtained in the paper.

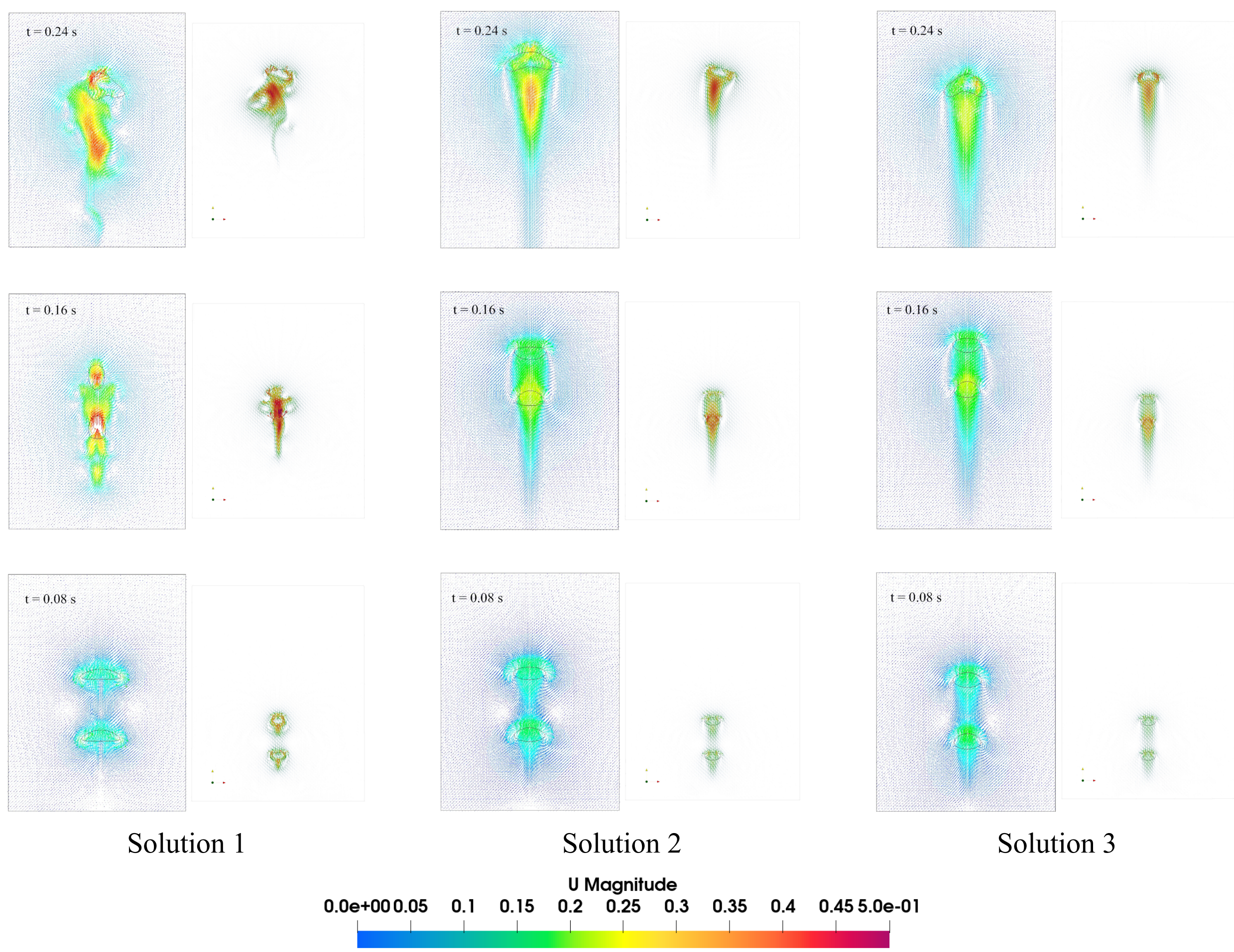


Figure 8: Comparison of instantaneous velocity field induced by two bubbles of diameter = 4mm rising in-line, as obtained in the paper vs results obtained in OpenFoam

4.3.2 Small bubble leading and large bubble trailing

fig. 9 depicts the trajectories of two bubbles; small leading bubble of diameter = 2mm and large trailing bubble of diameter = 4mm rising in-line. The bubbles did not coalesce in the time period considered for the study in solutions 1 and 2, but they coalesced in solution 3 due to the larger bubble rising faster. The instantaneous velocity values obtained in the OpenFoam simulations are in agreement with the values obtained in the paper

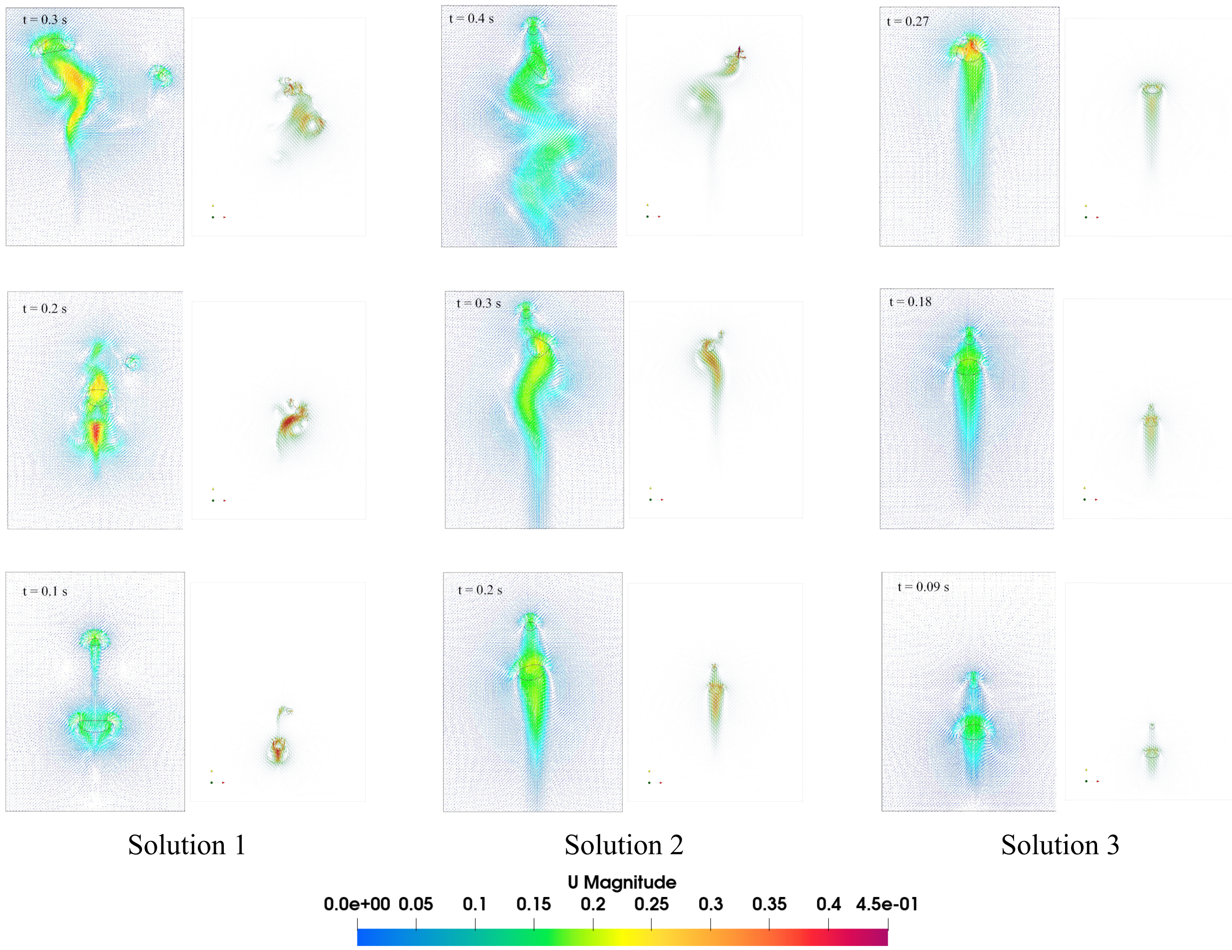


Figure 9: Comparison of instantaneous velocity field induced by two bubbles; small leading bubble diamter = 2mm & large trailing bubble diameter = 4mm rising in-line, as obtained in the paper vs results obtained in OpenFoam

4.3.3 Large bubble leading and small bubble trailing

fig. 10 depicts the trajectories of two bubbles; large leading bubble of diameter = 4mm and small trailing bubble of diameter = 2mm rising in-line. In none of the three solutions did the small bubble coalesce with the larger bubble, since the larger bubble was moving with a higher velocity. The quick movement of the large bubble away from the smaller bubble causes its region of wake to have decreasing effect on the smaller bubble with the increase in time. Here too the instantaneous velocity values obtained in the OpenFoam simulations are in good agreement with the results obtained in the paper.

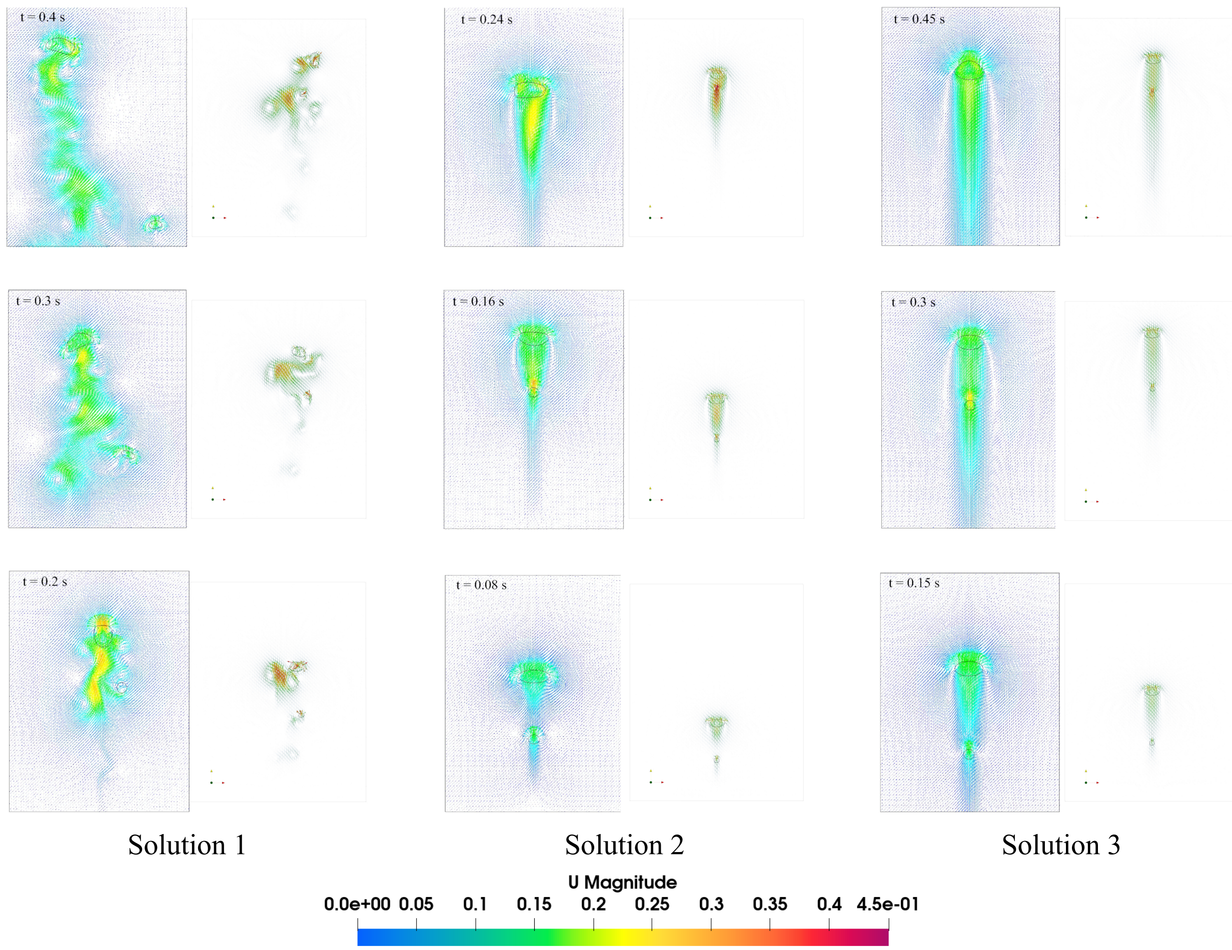


Figure 10: Comparison of instantaneous velocity field induced by two bubbles; large leading bubble diamter = 4mm & small trailing bubble diameter = 2mm rising in-line, as obtained in the paper vs results obtained in OpenFoam

5 References

[1] Abbassi, W., Besbes, S., Ben Aissia, H., & Champagne, J. Y. (2019). Study of the rise of a single/multiple bubbles in quiescent liquids using the VOF method. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, *41*(7), 272. https://doi.org/10.1007/s40430-019-1759-y
[2] Çengel, Y. A., Cimbala, J. M., & Turner, R. H. (2017). Fundamentals of Thermal-Fluid Sciences (5th Ed.). McGraw-Hill Education.
[3] https://openfoamwiki.net/index.php/InterFoam

Disclaimer

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.