Case study

Simulation of a 2D Rising Bubble Using the Volume of Fluid (VOF) Model in Ansys Fluent Software

Developed and curated by the Ansys Academic Program

Gautham Varma

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent®, a fluid simulation software

Summary

This study presents a numerical simulation of a single air bubble rising in different liquids (water, acetone and glycerin) column using the Volume of Fluid (VOF) model in Ansys Fluent Software. The transient multiphase flow is modeled to capture the bubble's deformation, rise velocity, and interaction with the surrounding liquid. Surface tension and buoyancy forces are included to accurately represent interphase dynamics. The results show that the bubble initially accelerates before reaching a stable terminal velocity, with formation into an ellipsoidal shape and finally breaks apart in the liquid

Table of Contents

1. Introduction

The study of gas bubble dynamics in a liquid medium has significant relevance in various engineering applications, including chemical reactors, bioreactors, and boiling processes. Understanding how bubbles rise, deform, and interact with the surrounding fluid is crucial for predicting multiphase flow behavior and optimizing process efficiency.

Computational Fluid Dynamics (CFD) provides a powerful approach for studying bubble dynamics without relying solely on experimental techniques, which can be expensive and difficult to visualize. Among the different multiphase modeling approaches, the Volume of Fluid (VOF) model is particularly effective for tracking the interphase between immiscible fluids, such as air and water, in transient simulations.

This case study demonstrates a 2D numerical simulation of a single gas bubble rising in a quiescent liquid column using Ansys Fluent software. The study highlights the setup, physics modeling, and key results to understand bubble deformation and terminal velocity behavior.

2. Problem Statement

The objective of this study is to simulate and analyze the rise of an air bubble in a column of water under the influence of gravity. The specific goals are:

1. To capture the transient motion and shape deformation of the bubble as it ascends through the liquid.
2. To predict the terminal rise velocity and compare it with theoretical or empirical correlations.
3. To observe the pressure and velocity fields in the liquid phase due to bubble motion.

The domain consists of a 2D water column with a single air bubble placed near the bottom. The simulation is assumed to be isothermal and involves two immiscible incompressible phases.

3. Theoretical Background

3.1 Understanding Multiphase Flow

Multiphase flow is defined as the simultaneous flow of materials with two or more distinct thermodynamic phases (i.e., gas, liquid, or solid). Unlike single-phase flow, where the fluid properties are continuous, multiphase flow is characterized by the presence of interphases that separate the phases.

The behavior of multiphase flow is significantly more complex than single phase simulations because of the presence of interphase which might include the exchange of mass, momentum and energy across these interphases.

3.1.1 Classification of multiphase flows

The main classification for multiphase flows happen based on the participating phases namely,

- *Gas-Liquid Flow* – The most common types (e.g.: boiling water, rising bubble in drinks or oil and gas pipelines)
- *Gas- Solid Flow* – Solid particles suspended in a gas stream mostly (e.g.: dust storms, soot in an exhaust of IC engines' factories etc.)
- *Liquid-Solid Flow* – Solid particles suspended and carried by liquids. (e.g.: slurry flows, sedimental flows, mineral exports in pipes etc.)
- *Liquid-Liquid Flow* – contains two immiscible liquids (e.g.: oil bubbles in water flows, treatment like liquid-liquid flows.)

The primary reason a bubble rises is Buoyancy. Because the air inside the bubble is much lighter than the surrounding water, the water pushes it upward. This is governed by Archimedes' Principle, which states the upward force (FB) equals the weight of the water the bubble pushes aside. Since this force is much stronger than the tiny downward pull of gravity (Fg) on the light gas, the bubble accelerates upward.

As the bubble speeds up, the water resists its movement, creating a downward Drag Force (FD). This drag increases the faster the bubble goes. Eventually, the upward push and the downward resistance balance out, and the bubble rises at a steady speed (terminal velocity). The motion is summarized by the force balance equation:

$$\mathrm{F_{net}} = \mathrm{F_B} + \mathrm{F_D} + \mathrm{F_g} \quad (1)$$

Where:

- $\mathrm{F_B} = \rho_{\mathrm{water}}.\mathrm{V}.\mathrm{g}$ (Upward Buoyancy)
- $\mathrm{F_D} = 1/2.\rho_{\mathrm{water}}.\mathrm{v}^2.\mathrm{C_d}.\mathrm{A}$ (Downward Drag)
- $\mathrm{F_g} = \mathrm{m_{bubble}}.\mathrm{g}$ (Downward Gravity)

With ρ is density of water, V is the volume of bubble, v is the velocity of the rising bubble, A is the cross-sectional area of the bubble, $\mathrm{C_d}$ is drag coefficient, m is mass of the air bubble, g is the acceleration due to gravity.

3.2 Volume of Fluid (VOF) method

The Volume of Fluid (VOF) method is one of the most widely used techniques for simulating two-phase or multiphase flows where the interphase between immiscible fluids (such as air and water) needs to be tracked accurately.

Originally introduced by Hirt and Nichols (1981)[1], the method has become a cornerstone in computational fluid dynamics (CFD) for applications like bubble dynamics, free-surface flows, droplet breakup, sloshing, and liquid jet simulations.

Unlike models that treat phases as interpenetrating continua (e.g., Eulerian or mixture models), the VOF approach is specifically designed to capture sharp interphases between fluids by solving for a volume fraction field that indicates how much of each fluid occupies a computational cell.

3.3 Fundamental Concept

The fundamental concept of the VOF method relies on defining a volume fraction (or "field") function (α) within each computational cell of the numerical grid. Here, α represents the fraction of the cell's volume occupied by a primary fluid (or the fluid of interest). The value of ranges from 0 to 1:

α=0: The cell is filled with the secondary fluid (the non-primary fluid).
α=1: The cell is filled with the primary fluid.
0<α<1: The cell contains the interphase between the two fluids.

This single scalar field is then transported throughout the domain by solving a conservation equation (usually a simple advection equation) that ensures the total volume of each fluid is conserved. Like every multiphase problem, the most important part of the calculation is the interphase tracking and reconstruction. As explained earlier, the value of field function (α) is used to mark the interphase cells, however, to accurately calculate the fluid fluxes, the shape and orientation of the interphase within the interphase cells also needs to be estimated. This process is defined as the interphase reconstruction. There are two main common reconstruction schemes which are as follows:

Simple Donor-Acceptor Scheme - This is a simple older computationally efficient scheme used in the VOF method. When calculating the amount of fluid passing from one cell (the donor) into the adjacent cell (the acceptor), the scheme assumes that all the flowing volume possesses the composition of the fluid present in the donor cell. This scheme completely ignores the shape or orientation of the actual interphase within the cell, looking at only the average value. The main advantage of this approach is the robustness, quickness and conservation of masses. However, the major drawback for the method is the significant numerical diffusion. This means the scheme will not be able to analyze the sharp boundary and it becomes smeared and blurry.

Piecewise Linear Interphase Calculation (PLIC) - This is an accurate method in VOF method to define clear boundaries between two phases. In each cell that contains the interphase, a clear line is also defined within the cell. The technique determines the interphase geometry by using the volume fraction in the neighboring cells to find the correct interphase orientation and then using the cell's own volume fraction to set the position of the resulting plane or line.

3.4 Advantages

» Good for capturing the sharp interphases between the immiscible fluids.
» Handles complex interphase topologies like breakup and coalescence.
» Computationally efficient compared to full interphase-tracking methods.
» Integrates easily with turbulence, heat transfer and phase change models.

3.5 Disadvantages

» Not an ideal method for multiple dispersed bubbles or droplets.
» Interphase smearing can occur if mesh resolution or advection schemes are inadequate.
» Difficult to model mass transfer or chemical reactions across the interphase.

4. Simulation Setup

The numerical investigation employs a two-dimensional planar computational domain representing a vertical fluid column of the size 70mm×600mm (L×H) as depicted in Figure 1. To guarantee precise resolution of the gas-liquid interphase, the domain is discretized into a structured mesh comprising 42,671 quadrilateral elements. This uniform grid density is designed to minimize numerical diffusion while maintaining high orthogonal quality, a critical requirement for convergence in multiphase calculations. The simulations are conducted using the transient, pressure-based solver within Ansys Fluent software, utilizing the Volume of Fluid (VOF) multiphase model. The Explicit VOF formulation is selected to strictly enforce interphase sharpness. Additionally, gravitational acceleration is applied in the negative Y-direction (-9.81 m/s) to initiate buoyancy-driven flow.

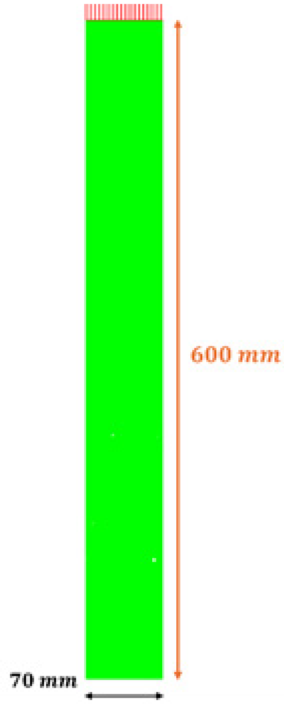

Figure 1: The domain of the Simulation setup.

To evaluate the influence of viscous forces on bubble hydrodynamics, three distinct continuous phases—Water, Acetone, and Glycerin—were selected, offering a significant range of dynamic viscosities spanning from the inertial to the viscous regime (detailed in Table 1). The numerical solution is controlled using the SIMPLE scheme for pressure-velocity coupling, providing a robust baseline for the pressure-based formulation. Spatial discretization employs the PRESTO! scheme for pressure, which is optimal for flows with high hydrostatic gradients, and the Second-Order Upwind scheme for momentum equations to enhance accuracy.

Table 1 : The Properties of the three liquids used in this simulation.

Properties (at $25^\circ\mathrm{C}$)	Water	Acetone	Glycerin
Density (ρ [kgm3])	997	791	1259.9
Viscosity (μ [kgm s])	0.00089	0.000331	0.799
Surface Tension (N/m)	0.072 [2]	0.024	0.0634 [3]

Temporal advancement is achieved through a fixed time-stepping approach rather than adaptive stepping, ensuring consistent temporal resolution across all test cases. The solution is initialized with the domain fully saturated by the liquid phase, after which a circular air bubble is introduced at t=0s via region patching at the base of the domain.

5. Results

The simulation of an air bubble rising in glycerin, as illustrated by the velocity Figure 2 and Volume Fraction Figure 3 contours, corresponds to a flow regime strongly governed by viscous forces. Over the two-second simulation period, the bubble retains a coherent, nearly spherical to mildly ellipsoidal shape, with no evidence of interfacial breakup or significant deformation. This morphological stability arises from the high dynamic viscosity of glycerin, which effectively damps inertial and surface instabilities that would otherwise distort the interphase in lower-viscosity fluids.

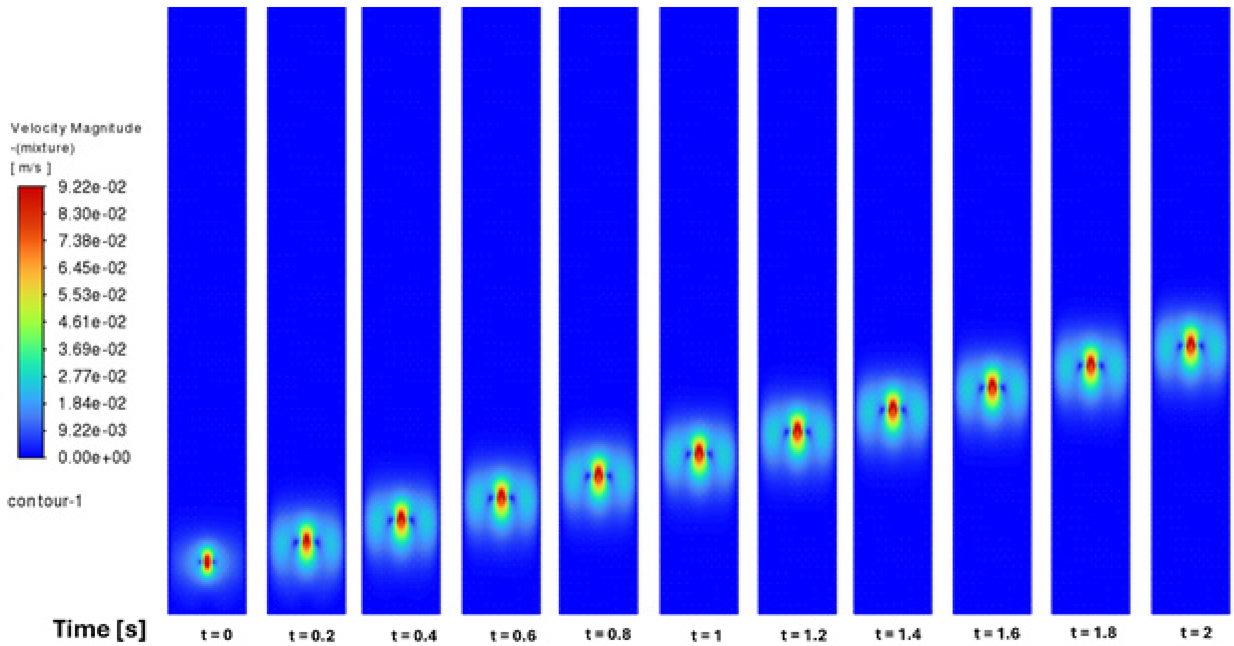

Figure 2: The Velocity contour plot shows the profile of rising air bubble in glycerin.

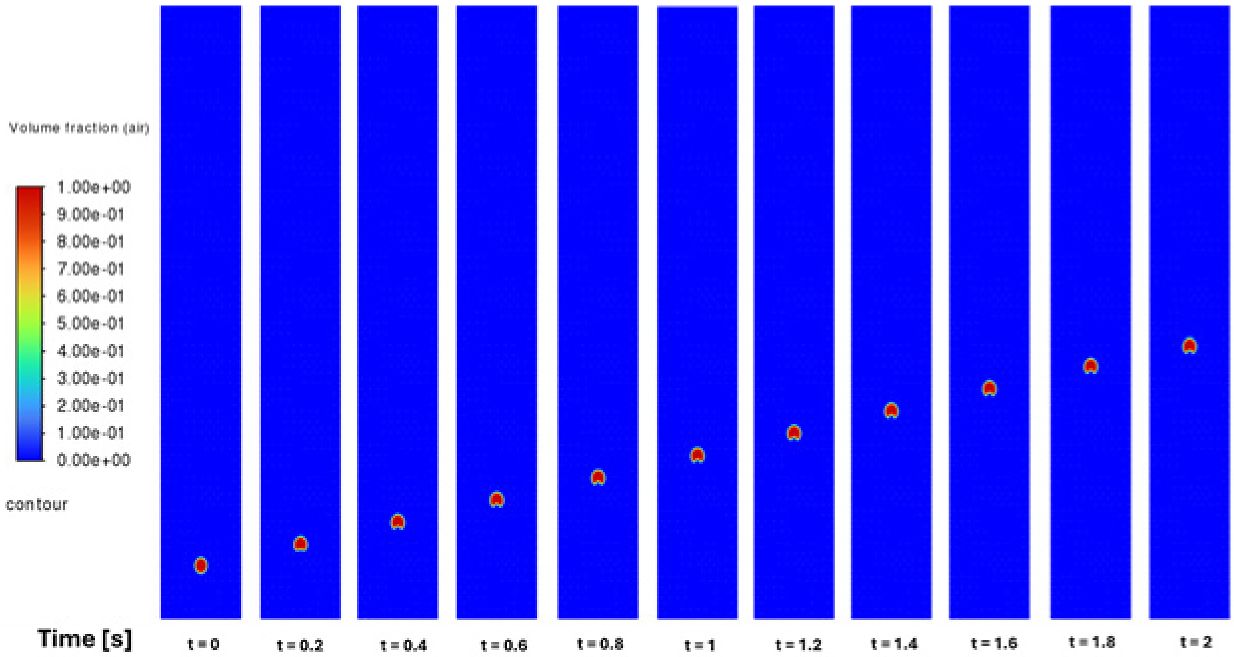

Figure 3: The Volume fraction contour plot shows the profile of rising air bubble in glycerin.

The velocity field (Figure 2) indicates a maximum bubble rise velocity of approximately 0.092 m/s, substantially lower than that observed in the other test cases. This reduced ascent rate is characteristic of low-Reynolds-number flows (approx. equal to 6 for the air bubble), where viscous drag dominates over inertial effects. The surrounding flow remains laminar and symmetric, with a smooth wake structure and no signs of vortex shedding. Consequently, the bubble follows a straight, vertically aligned trajectory throughout the simulation.

In contrast to glycerin, the air-water system exhibits high Reynolds number behavior where inertial forces overcome viscous damping Figure 4. Initially spherical, the bubble rapidly accelerates and deforms into a characteristic "skirted" or mushroom-cap geometry (Figure 5, t=0.2s) due to the non-uniform dynamic pressure distribution across its surface. The Volume Fraction contours (Figure 5) clearly capture the development of Kelvin-Helmholtz instabilities along the bubble's skirt, leading to the shedding of "satellite" bubbles at the trailing edge. The rising velocity is substantially higher, peaking at 0.356 m/s. However, the trajectory is non-linear; the bubble exhibits a distinct wobbling motion caused by a turbulent wake and vortex shedding.

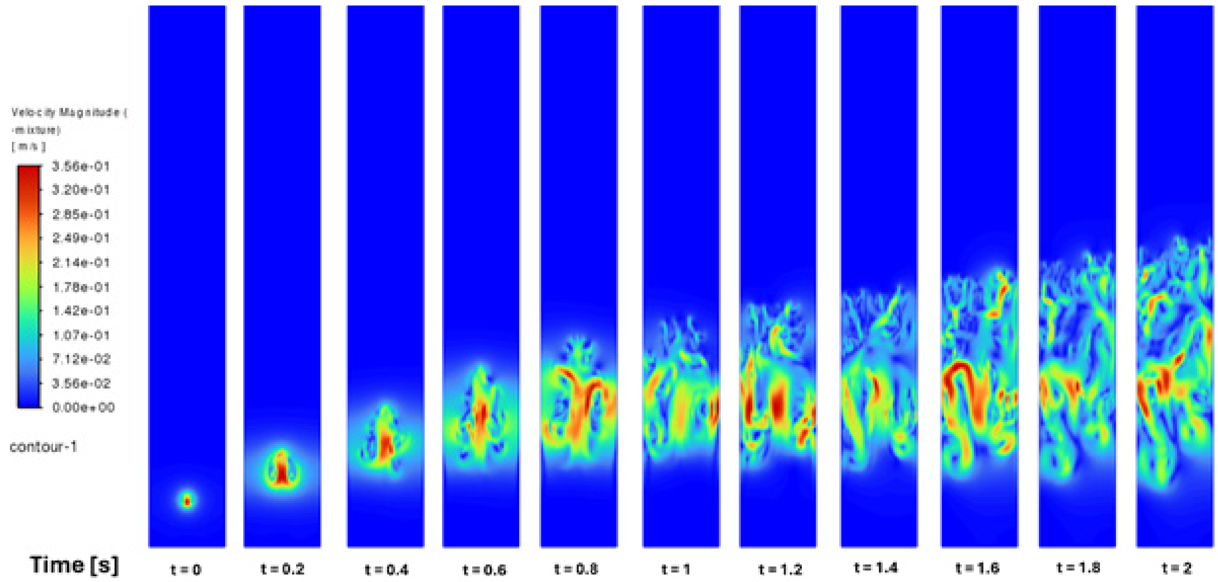

Figure 4: The Velocity contours of air bubble inside water solution.

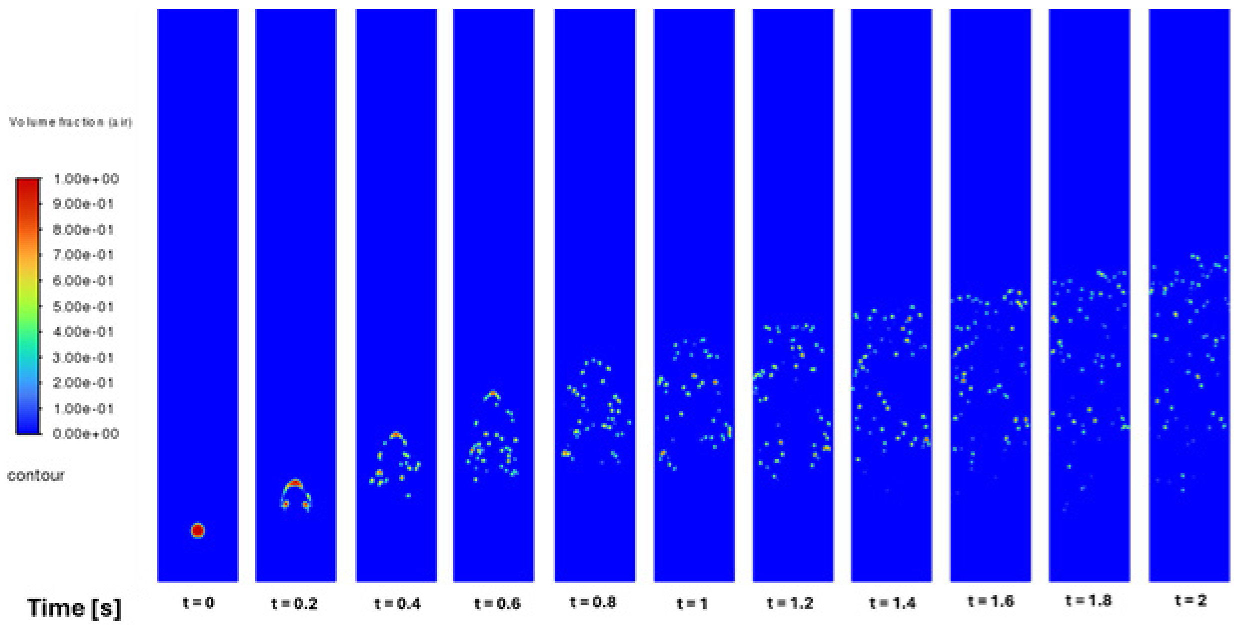

Figure 5: The Volume fraction contour of air showing the movement and shredding of bubble inside the water .

The air-acetone simulation demonstrates the critical role of surface tension in multiphase flows. While the rise velocity (0.357 m/s, Figure 6), and general flow regime are similar to that of water due to comparable low viscosities, the topological evolution of the interphase is markedly different. Acetone possesses a significantly lower surface tension (σ=0.024 N/m) than water. Consequently, the Weber number, which represents the ratio of inertial forces to surface tension forces, is higher for the acetone case. This reduced interfacial strength allows hydrodynamic shear forces to tear the bubble interphase more aggressively. The results show extensive fragmentation and atomization, where the main bubble breaks down into a chaotic swarm of fine micro-bubbles (Figure 7). The wake is highly turbulent, and the VOF method captures a wider dispersion of the gas phase across the domain width compared to the water case.

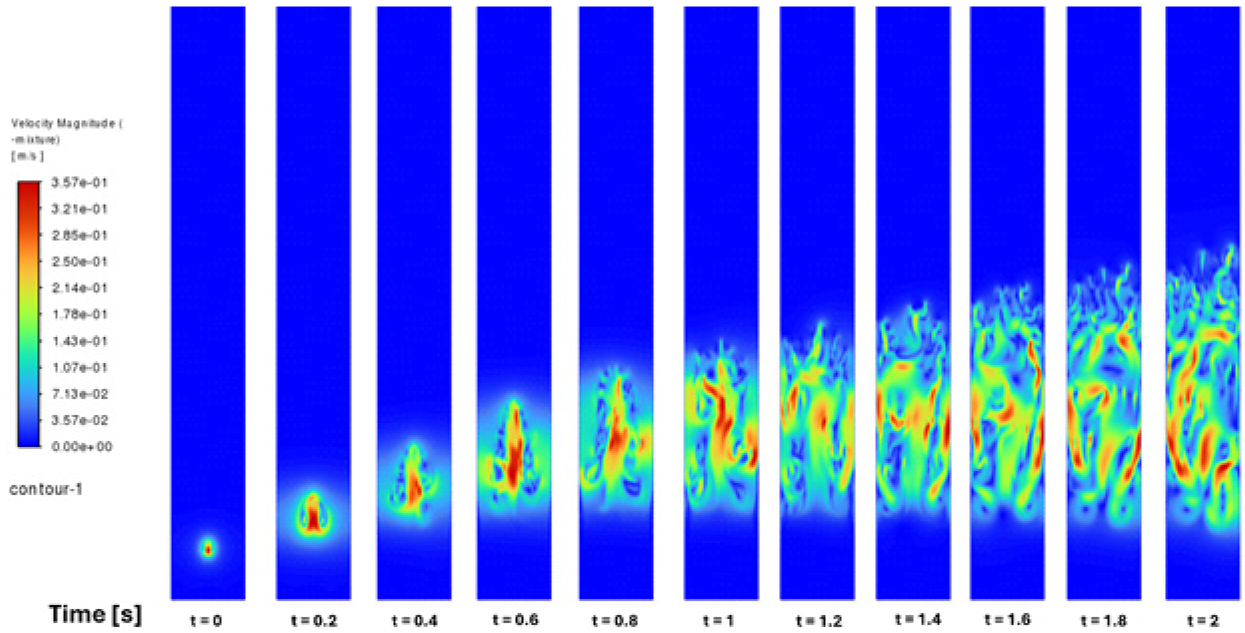

Figure 6: Velocity Profile of the air bubble inside acetone solution showing the wider spread of bubbles due to atomization.

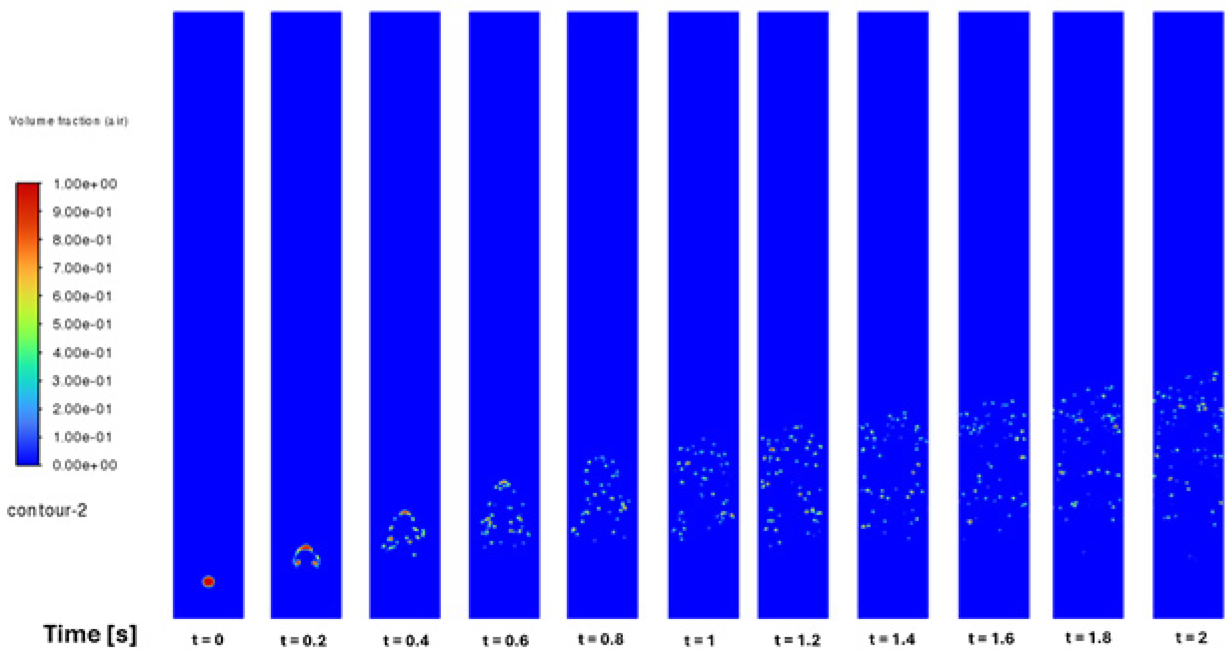

Figure 7: The Volume Fraction contour of air bubble inside acetone solution showing the atomization of particles.

The results validate the Ansys Fluent Volume of Fluid (VOF) model as an excellent tool for capturing the diverse physics of buoyancy-driven flows. The model's strength lies in its ability to handle topological changes without the need for mesh motion or re-meshing. In the glycerin case, the explicit VOF formulation with the Geo-Reconstruct scheme successfully resolved a sharp, crisp interphase without numerical diffusion (smearing), which is essential for accurate drag prediction in laminar flows. In the acetone and water cases, the solver accurately captured complex topological events, such as interphase tearing, coalescence, and the formation of sub-grid droplets. The ability to resolve the fine "mist" of bubbles in the acetone simulation highlights the model's sensitivity to surface tension parameters and its capability to conserve mass even during chaotic fragmentation.

6. Conclusions

The comparative analysis demonstrates that fluid viscosity is the dominant factor governing bubble rise velocity and trajectory stability. In highly viscous glycerin, the bubble ascends at nearly one-fourth the velocity observed in water and acetone, reflecting the strong damping effect of viscous forces on buoyancy-driven motion. In contrast, surface tension primarily controls the integrity of the gas–liquid interphase. The comparatively low surface tension of acetone promotes pronounced interfacial instability, resulting in extensive bubble fragmentation, whereas the higher surface tension of water preserves a largely coherent bubble structure.

Across this broad spectrum of flow regimes, from viscous, laminar ascent to inertia-dominated, highly transient breakup, the Volume of Fluid (VOF) method demonstrates strong robustness and reliability. It successfully captures both the steady, stable bubble dynamics in high-viscosity fluids and the complex, unsteady interfacial evolution characteristic of low-viscosity, low-surface-tension systems.

7. References

[1] Hirt, Cyril W., and Billy D. Nichols. “Volume of fluid (VOF) method for the dynamics of free boundaries.” Journal of computational physics 39.1 (1981): 201-225.

[2] A. P. P. S. A. K. Allan H Harvey, NIST/ASME steam properties, 2008.

[3] Cleaning Institute, “Physical properties of Glycerin and its solutions,” [Online]. Available: https://www.cleaninginstitute.org/sites/default/files/research-pdfs/Physical_properties_of_glycerine_and_its_solutions.pdf.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found using this link.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com