fluids

Editorial

Computational Fluid Dynamics Modeling and Experiments of Two-Phase Flows †

Van-Tu Nguyen [1,*] and Hemant J. Sagar [2]

[1] School of Mechanical Engineering, Pusan National University, Busan 46241, Republic of Korea
[2] Department of Hydro and Renewable Energy, Indian Institute of Technology (IIT), Roorkee 247667, Uttarakhand, India; hemant.sagar@hre.iitr.ac.in
* Correspondence: vantunguyen@pusan.ac.kr

† This paper is part of the special issue, Numerical Modeling and Experimental Studies of Two-Phase Flows.

Citation: Nguyen, V.-T.; Sagar, H.J. Computational Fluid Dynamics Modeling and Experiments of Two-Phase Flows. *Fluids* **2024**, *9*, 207. https://doi.org/10.3390/fluids9090207

Received: 27 August 2024
Accepted: 30 August 2024
Published: 3 September 2024

Two-phase flows are prevalent in natural phenomena, as well as a wide range of marine engineering and industrial applications. However, the study of two-phase systems has been limited due to the added complications of the two-phase interface, where mass transfer causes complex flow behaviors. These flows involve the interaction of two distinct phases within a system, leading to highly nonlinear dynamics. Some examples of these flows include free surface flows interacting with marine and offshore structures; cavitation; steam and water flow in power plants; oil and gas transportation in pipelines; boiling and condensation in heat exchangers; and other natural occurrences. Experimental studies of two-phase flows face significant difficulties—they are often expensive, time-consuming, and complex to set up, particularly when replicating extreme conditions or achieving detailed visualizations. Despite these challenges, experiments are essential for validating and confirming computational fluid dynamics (CFD) results. While CFD methods are crucial for simulating and analyzing two-phase flows due to their flexibility and cost-effectiveness, experimental data are essential for ensuring the accuracy and reliability of these numerical models. This Special Issue in Fluids, entitled "Numerical Modeling and Experimental Studies of Two-Phase Flows," focuses on recent advances in both numerical and experimental modeling of two-phase flows, providing deeper insights into the fundamental and physical aspects of these flows across several fields, including engineering and industry.

Bubble dynamics are a crucial phenomenon in fluid mechanics, impacting both natural processes and engineered systems. The study of bubble dynamics encompasses the formation, growth, oscillation, and collapse of bubbles within a fluid. These processes are influenced by factors such as pressure variations, surface tension, and the presence of interfaces between fluids [1]. The toroidal model and the ring shedding approach for toroidal bubble dynamics were numerically analyzed to address the discontinuous pressure field on the bubble interface, simulating behavior in terms of bubble geometry, internal gas pressure, and shock wave propagation [2]. Building on the framework of smoothed particle hydrodynamics (SPH) and incorporating the van der Waals (VDW) equation of state, the deformation and the collapse of a heated vapor bubble near a solid boundary were examined as the bubble ascended from the bottom surface [3]. Cavitation is a phenomenon that occurs when the pressure in a liquid falls below its vapor pressure, resulting in the formation of vapor-filled bubbles or cavities. These bubbles can rapidly expand and collapse, producing extreme local pressures and temperatures. Cavitation commonly arises in hydraulic systems and fluid machinery, including turbines, impellers, nozzles, and underwater propulsion systems. Although cavitation is often associated with equipment damage, such as material erosion and surface pitting, it can also be utilized in advantageous ways, such as in ultrasonic cleaning, water treatment, and medical applications such as lithotripsy [1]. The behavior of a cavitation bubble near a rigid conical surface is explored through a combination of numerical simulations and experimental observations, revealing key phenomena such as shock wave emissions, liquid jet formations, and localized high-pressure regions. In the experimental setup, a single cavitation bubble is produced using a pulsed laser, with its dynamics being recorded using a high-speed camera at 100,000 frames per second. For the numerical analysis, a compressible two-phase flow model that incorporates phase transitions and thermal effects is utilized, implemented in the OpenFOAM framework [4]. Utilizing high-resolution two-dimensional Particle Imaging Velocimetry, the internal flow dynamics of enlarged transparent models of multi-hole injectors were investigated. These measurements were aimed at comprehending the mechanisms behind bulk cavitation formation and their relationship to the injector's flow characteristics, subsequently providing insights into the flow field behaviors within the injector's internal structure [5]. The periodic behavior and formation mechanisms of cavitation clouds in a submerged water jet from an orifice nozzle were analyzed, where high-speed camera imaging and flow simulations reveal that pairs of ring-like clouds, consisting of a leading and a subsequent cloud, are periodically shed downstream [6]. The findings reveal that the leading cloud separates due to a shear vortex at the nozzle exit, while the subsequent cloud is released via a re-entrant jet following the collapse of a fully extended cavity. The impact of cavitation on the hydrodynamic properties of a circular cylinder in diverse cavitating flows was studied experimentally. The hydrodynamic forces acting on the cylinder were measured using a load cell, while a high-speed camera was employed to capture the cavitation dynamics occurring behind the cylinder. Subsequently, the cavitation behavior around the cylinder was analyzed for various cavitating regimes, including the inception of cavitation, partial cavitation, and cloud cavitation. [7]. During cavitation bubble collapse, shock waves and high-speed microjets, at speeds of up to thousands of meters per second, create high local energy concentrations, with temperatures exceeding tens of thousands of Kelvin and pressures reaching gigapascal levels. This extreme environment may lead to behaviors similar to those of supercritical fluids, including the release of non-condensable gasses, plasma formation, and chemical reactions. Both current computational fluid dynamics (CFD) models and experimental methods struggle to accurately predict these phenomena. Numerically and experimentally investigating the effect of non-condensable gas on the dynamics, pressure, and temperature could be challenging in the future. Understanding bubble dynamics is essential for numerous applications, from improving industrial processes to advancing biomedical technologies.

Flow within channels is a fundamental concept in fluid mechanics, describing the movement of fluids through confined pathways, such as pipes, ducts, and open channels. A comprehensive review of the literature on CHF in vertical downflow channels was conducted [8]. The behavior of boiling under downflow conditions in vertical channels, which is relevant to applications such as steam generators, power plants, and industrial cooling systems, has been thoroughly reviewed. A detailed comparison of existing correlations with experimental data and the prediction of critical heat flux (CHF) was presented, providing insights into flow stability and thermal performance. Additionally, the effect of a transverse magnetic field on a two-phase stratified flow in both horizontal and inclined channels was investigated [9]. The study in question explored how the magnetic field influences laminar stratified flows, particularly when the more dense, electrically conductive liquid occupies the lower layer, while the upper fluid acts as an electrical insulator. The introduction of a transverse magnetic field adds another dimension to flow control, particularly in systems with electrically conductive fluids, offering potential for enhanced flow stability. Meanwhile, the application of ultrasonic techniques for the real-time monitoring of bubbly flows presents promising opportunities for improving diagnostic capabilities in both industrial and research settings. In another investigation, the use of ultrasonic techniques to monitor bubbly flow and to determine bubble density in a water column was documented [10]. The quantity of bubbles was assessed by analyzing the performance of the positive displacement pump that was employed for air injection. The findings indicated that bubble density in the water column can be effectively monitored using the phase spectrum of the loss coefficient. Further research is warranted to refine these models and techniques, particularly in scaling them up for larger, more complex systems.

Two-phase flow research addresses significant real-world challenges such as understanding the breaking of waves and the overtopping of coastal structures, as well as analyzing the interactions of moving ships with extreme waves and green water on decks. Additionally, applications such as spray cooling, two-phase heat transfer, hydrodynamic cavitation, and dynamic bubble processes benefit from these advances, ultimately enhancing critical heat flux and the system's reliability. Despite the advantages of computational fluid dynamics (CFD) in studying two-phase flows, numerous challenges persist. Accurately modeling phase interactions, handling complex boundary conditions, ensuring numerical stability, and managing significant computational demands are some of the ongoing issues. Continuous advancements in computational methods, high-performance computing, and validation techniques are essential to fully leverage the potential of CFD in two-phase flow studies [11]. The integration of machine learning (ML) and deep learning (DL) into CFD represents a major step forward in optimizing fluid dynamics simulations. Traditional CFD models, while powerful, can be computationally expensive and time-consuming, especially when dealing with complex two-phase flows. As highlighted in [12], the industrial market is becoming increasingly competitive, pushing companies to adopt advanced technologies to gain a strategic advantage. One key resource is simulation, which plays an essential role in Industry 4.0, particularly in layout reconfiguration, to enable flexible product customization and to optimize manufacturing processes. In this context, computational fluid dynamics (CFD) simulations offer a substantial competitive advantage for smart factories by leveraging emerging technologies. In addition to the continued development of CFD methods and modeling techniques for two-phase flows, recent years have seen the rise of a transformative technology (ML and DL). ML and DL techniques can provide faster predictions, potentially reducing computational costs while maintaining accuracy. These techniques are reshaping various domains, and their impact on CFD is expected to be significant. A noteworthy application involves using DL methods to predict particle concentration in gas–solid two-phase flows [13]. This study compares the effectiveness of three approaches—Back-Propagation Neural Networks (BPNNs), Recurrent Neural Networks (RNNs), and Long Short-Term Memory (LSTM) networks—in analyzing gas–solid two-phase flow data. Seven key parameters, including temperature, humidity, upstream and downstream sensor signals, delay, pressure difference, and particle concentration, were utilized to construct the dataset. The comparison of different neural network architectures demonstrates the versatility of these methods in handling multifaceted datasets, enabling more accurate predictions of complex flow behaviors such as particle concentration in gas–solid systems. This advancement opens up new possibilities for real-time monitoring and control in industrial applications.

Another study, which explores the use of machine learning for analyzing and estimating pressure drop in two-phase flow dynamics within smooth tubes, has been documented [14]. This research begins with experimental measurements of pressure drop for a water–air mixture across various flow conditions in horizontally oriented smooth tubes. ML techniques are then applied to predict pressure drop values using dimensionless parameters derived from the experimental data. Feature selection methods are employed to identify crucial features, which aids in better understanding the underlying physical mechanisms and improving the accuracy of the models. Furthermore, a genetic algorithm is employed to optimize the selection of the machine learning model and its tuning parameters. As a result, the optimized pipeline achieves a low mean absolute percentage error on both the validation and test datasets. These ongoing studies on combining CFD and ML or DL approaches will continue to provide crucial insights and solutions for both natural and engineered systems, particularly in maritime contexts where efficient and innovative designs are paramount.

Acknowledgments: The Guest Editors would like to thank all contributing authors and reviewers for their invaluable efforts in enhancing the quality of this Special Issue. We are particularly grateful to the editors of *Fluids* for their steadfast support and encouragement throughout the editing process. The contributions of the anonymous reviewers were essential, and their dedication is greatly appreciated.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Nguyen, V.-T.; Sagar, H.J.; Moctar, O.e.; Park, W.-G. Understanding cavitation bubble collapse and rebound near a solid wall. *Int. J. Mech. Sci.* **2024**, *278*, 109473. [CrossRef]
2. Han, L.; Zhang, T.; Yang, D.; Han, R.; Li, S. Comparison of Vortex Cut and Vortex Ring Models for Toroidal Bubble Dynamics in Underwater Explosions. *Fluids* **2023**, *8*, 131. [CrossRef]
3. Chen, Y.; Wang, Q.; Xiong, H.; Qian, L. Vapor Bubble Deformation and Collapse near Free Surface. *Fluids* **2023**, *8*, 187. [CrossRef]
4. Yin, J.; Zhang, Y.; Gong, D.; Tian, L.; Du, X. Dynamics of a Laser-Induced Cavitation Bubble near a Cone: An Experimental and Numerical Study. *Fluids* **2023**, *8*, 220. [CrossRef]
5. Kolokotronis, D.; Sahu, S.; Hardalupas, Y.; Taylor, A.M.K.P.; Arioka, A. Bulk Cavitation in Model Gasoline Injectors and Their Correlation with the Instantaneous Liquid Flow Field. *Fluids* **2023**, *8*, 214. [CrossRef]
6. Onishi, T.; Li, K.; Ji, H.; Peng, G. Shedding of Cavitation Clouds in an Orifice Nozzle. *Fluids* **2024**, *9*, 156. [CrossRef]
7. Lin, Y.; Kadivar, E.; el Moctar, O. Experimental Study of the Cavitation Effects on Hydrodynamic Behavior of a Circular Cylinder at Different Cavitation Regimes. *Fluids* **2023**, *8*, 162. [CrossRef]
8. Shah, M.M. Prediction of Critical Heat Flux during Downflow in Fully Heated Vertical Channels. *Fluids* **2024**, *9*, 79. [CrossRef]
9. Parfenov, A.; Gelfgat, A.; Ullmann, A.; Brauner, N. Hartmann Flow of Two-Layered Fluids in Horizontal and Inclined Channels. *Fluids* **2024**, *9*, 129. [CrossRef]
10. Franco, E.E.; Henao Santa, S.; Cabrera, J.J.; Laín, S. Air Flow Monitoring in a Bubble Column Using Ultrasonic Spectrometry. *Fluids* **2024**, *9*, 163. [CrossRef]
11. Nguyen, V.-T.; Park, W.-G. A Review of Preconditioning and Artificial Compressibility Dual-Time Navier–Stokes Solvers for Multiphase Flows. *Fluids* **2023**, *8*, 100. [CrossRef]
12. Silvestri, L. CFD modeling in Industry 4.0: New perspectives for smart factories. *Procedia Comput. Sci.* **2021**, *180*, 381–387. [CrossRef]
13. Wang, Z.; Yan, B.; Wang, H. Application of Deep Learning in Predicting Particle Concentration of Gas–Solid Two-Phase Flow. *Fluids* **2024**, *9*, 59. [CrossRef]
14. Bolourchifard, F.; Ardam, K.; Dadras Javan, F.; Najafi, B.; Vega Penichet Domecq, P.; Rinaldi, F.; Colombo, L.P.M. Pressure Drop Estimation of Two-Phase Adiabatic Flows in Smooth Tubes: Development of Machine Learning-Based Pipelines. *Fluids* **2024**, *9*, 181. [CrossRef]