Laurila, Erkki; Izbassarov, Daulet; Järvinen, Mika; Vuorinen, Ville

Numerical study of bubbly flow in a swirl atomizer

Published in:
PHYSICS OF FLUIDS

DOI:
10.1063/5.0028963

Published: 02/12/2020

Document Version
Peer reviewed version

Published under the following license:
Unspecified

Please cite the original version:
Laurila, E., Izbassarov, D., Järvinen, M., & Vuorinen, V. (2020). Numerical study of bubbly flow in a swirl atomizer. PHYSICS OF FLUIDS, 32(12), [122104]. https://doi.org/10.1063/5.0028963

This material is protected by copyright and other intellectual property rights, and duplication or sale of all or part of any of the repository collections is not permitted, except that material may be duplicated by you for your research use or educational purposes in electronic or print form. You must obtain permission for any other use. Electronic or print copies may not be offered, whether for sale or otherwise to anyone who is not an authorised user.
Numerical study of bubbly flow in a swirl atomizer

Erkki Laurila,1,a) Daulet Izbassarov,1 Mika Järvinen,1 and Ville Vuorinen1

Department of Mechanical Engineering, Aalto University, FI-00076 AALTO, Finland

(Dated: 27 October 2020)

In the present work, we extend our previous research on swirl nozzles by introducing bubbles at the nozzle inlet. A large-scale hollow cone pressure-swirl atomizer is studied using scale-resolving simulations. The present flow conditions target a Reynolds number range of $600 \leq Re \leq 910$ and gas-to-total volumetric flow rate ratios between $0.07 \leq \beta \leq 0.33$ with $\beta = 0$ as an experimental and computational reference. The computational setup has relevance to high-viscosity bio-fuel injection processes. Flow rate ratio and bubble diameter sweeps are carried out to study their effect on the inner-nozzle flow and the liquid film characteristics outside the nozzle.

The present flow system is shown to pose highly versatile physics including bubble coalescence, bubble-vortex interaction, and faster liquid film destabilization relative to $\beta = 0$ case. The main results are as follows. (1) $\beta$ is found to have a significant effect on the bimodal bubble volume probability density function (PDF) inside the swirl chamber. Also, the total resolved interfacial area of the near-orifice liquid film increases with $\beta$. (2) At representative value of $\beta = 0.2$, the exact bubble size at the inlet is observed to have only a minor effect on the swirl chamber flow and liquid film characteristics. (3) The bubble-free ($\beta = 0$) and bubbly ($\beta > 0$) flows differ in terms of effective gas core diameter, core intermittency features, and spray uniformity. The quantitative analysis implies that bubble inclusion at the inlet affects global liquid film characteristics with relevance to atomization.

a)Author to whom correspondence should be addressed: erkki.laurila@aalto.fi

For more information, please cite this article as DOI: 10.1063/5.0028963
I. INTRODUCTION

Here, two-phase gas-liquid flow is studied inside a large-scale pressure-swirl atomizer used for injection of high-viscosity, bio-mass based liquid fuels in boiler applications. Studying in-nozzle two-phase flows has relevance for various scenarios related to injector operation. For example, the injected liquid can be aerated with gas in order to enhance atomization of the subsequent liquid spray (effervescent atomization). In addition, the gas (vapor) may be generated by boiling process inside the nozzle. For example, modern recovery boilers are often operated in conditions susceptible for flash boiling. In addition to having a strong impact on the spray quality, flashing affects the in-nozzle flow as significant vapor generation may occur already inside the nozzle (approximate void fraction as high as 50–70% at nozzle exit). Despite the strong influence on the spray characteristics, the effect of the gas presence on the nozzle flow has thus far received little attention.

Pressure-swirl atomizers utilize swirling motion to enhance atomization. In a swirl nozzle, the flow enters a swirl chamber through tangential or axial (vaned) inlet ports which creates a strong vortex characterized by the formation of a gaseous core in typical operating conditions. Customarily, swirl nozzles are operated with purely single-phase liquid feeds. The spray undergoes different transitional modes before reaching a fully developed hollow cone flow regime when the inlet velocity, and therefore the swirl strength, is increased. The hollow cone mode leads to increased spray angle, discharge velocity and amount of air-to-liquid interaction which, in turn, enhance atomization. For review of research on swirl atomizers, see Vijay, Moorthi, and Manivannan and Kang et al.

Significant body of literature on effervescent atomizers highlights the importance of the structure of the inner-nozzle two-phase flow for the spray characteristics. Ochowiak, Broniarz-Press, and Rozanski measured the discharge coefficient of effervescent atomizers and discussed the influence of geometry and flow regime. They place the transition between bubbly and annular flow at a gas-to-liquid mass flow ratio (GLR) of 0.07. Similarly, Jedelsky and Jicha place this transition (steady vs. unsteady operation) at GLR = 0.06. The conditions in the present study correspond to a low GLR range of $\mathcal{O}(10^{-4})$. Huang, Wang, and Liao used high-speed photography and optical measurements to characterize flow regimes inside an effervescent atomizer. They also report how the in-nozzle two-phase flow structures affect the Sauter mean diameter (SMD) and other spray characteristics.
Correspondingly, Lin, Kennedy, and Jackson\textsuperscript{11} concluded that the aerated-liquid spray is strongly related to the structure of the internal two-phase flow inside a small-scale atomizer (orifice diameter $d_o \sim 2 \text{ mm}$) by testing out various discharge passage lengths and converging angles. Unsteady effects in effervescent atomizers have been studied for example by Sen \textit{et al.}\textsuperscript{12} and Hong, Fleck, and Nobes\textsuperscript{13}.

Computational studies of effervescent atomizers have mainly concentrated on the nozzle exit region. Sun \textit{et al.}\textsuperscript{14,15} investigated the internal flow in a converging effervescent nozzle exit region with 3d simulations. The internal flow pattern was noted to affect the fluctuation characteristics of the spray. Mousavi and Dolatabadi\textsuperscript{16} conducted 3d large-eddy simulations of both internal and external flow in an effervescent atomizer and demonstrated decreasing breakup lengths and increasing spray cone angles with increasing GLR. Also other studies on atomizer internal flow have been carried out in 2d\textsuperscript{17–19} and 3d\textsuperscript{20}. All of the aforementioned studies utilized the volume-of-fluid (VOF) method for interface capturing. In addition, Eulerian-Eulerian two-fluid models have been used to study flows in effervescent atomizers\textsuperscript{21,22}.

In the context of pressure-swirl atomizers, the interaction of the main vortex with the two-phase flow is important. Bubbles in a swirling flow start to migrate towards the center of rotation due to centrifugal force which is larger for liquid\textsuperscript{23,24}. Maneshian, Javadi, and Taebi Rahni\textsuperscript{23} studied numerically the evolution of single bubbles in 3d under the combined effect of rotating flow and gravitational acceleration for a range of Morton and Bond numbers. Similarly, Hsiao, Ma, and Chahine\textsuperscript{25} studied a swirl separation process in different gravity environments. Migration towards center of rotation was also observed by Van Nierop \textit{et al.}\textsuperscript{24}, who studied small air bubbles in rotating flow in order to determine the drag and lift coefficients of the bubbles. Effervescent atomization in swirl nozzle context has been investigated by Ochowiak\textsuperscript{26}. They studied the discharge coefficient ($C_d$) of a small-scale ($d_o \sim 2 \text{ mm}$) effervescent-swirl atomizer with various fluids and orifice shapes. The nozzle geometry (orifice shape and type) was noted to greatly affect $C_d$. They also demonstrated diminishing $C_d$ with GLR, while noting that at higher GLR the shape of the orifice did not play an important role.

Recently, Laurila \textit{et al.}\textsuperscript{27} investigated a large-scale pressure-swirl atomizer ($d_o \sim 2 \text{ cm}$) with significant asymmetric features in contrast to the more commonly studied small-scale symmetric nozzle types. Numerical simulations were conducted at a Reynolds number range
of $420 \leq Re \leq 5300$ and several flow modes were identified. S-shaped, transitional and fully developed hollow cone liquid films were observed at $Re = 420$, $830$ and $\geq 1660$, respectively. Later, the work was extended by conducting both experimental and numerical investigations at a focused Reynolds number range ($600 \leq Re \leq 910$). At this range, the flow was found to be in a transitional state without a fully developed air core consistent with the previous findings.

In the present work, we extend our previous studies by introducing bubbly flow at the inlet of the nozzle. Understanding the characteristics of the in-nozzle two-phase flow has relevance for injector operation and subsequent spray formation. The literature highlights the importance of nozzle geometry in the case of effervescent atomization. However, there are only a few studies on internal flow in effervescent-swirl atomizers and especially on large-scale designs such as the current geometry. The presently studied scenario features complex flow physics including developing two-phase pipe flow with bubble coalescence, interaction of gas structures with swirling flow and destabilization of the ejected liquid film by bubbles. The main goals of this study are to:

1. Relate and compare the present bubbly flow scenarios to the corresponding cases without added gas.
2. Investigate the effect of gas-to-total volumetric flow rate ratio on the general flow characteristics.
3. Study the effect of inlet bubble diameter on the flow dynamics including two-phase flow structures.

II. METHODS

A. Governing equations and numerical methods

Here, the same numerical method as in our previous publications is utilized. The volume-of-fluid method is used for the time dependent solution of the two-phase flow. In VOF, the phases are identified by the indicator field, $\alpha$, which obtains a value of one in the liquid and zero in the gas phase, i.e. $\alpha = \alpha_l$. Based on the incompressible Navier-Stokes
equations, the flow dynamics are governed by the advection equation for the indicator field
\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot \alpha \mathbf{u} = 0,
\]
and the momentum equation which incorporates the effects of the gravitational, viscous and surface tension forces
\[
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \rho \mathbf{g} + \nabla \cdot \left[ \mu \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) \right] + \sigma \kappa \nabla \alpha.
\]
Above, \( \mathbf{u} \), \( \rho \), \( p \), \( \mu \), \( \sigma \) and \( \kappa \) are the velocity, density, pressure, viscosity, surface tension and curvature, respectively. The density and viscosity are mixture quantities and they are calculated as \( \rho = \alpha \rho_l + (1-\alpha)\rho_g \) and \( \mu = \alpha \mu_l + (1-\alpha)\mu_g \) based on the phase-specific values (\( l \) for liquid, \( g \) for gas). In Eq. 2, the surface tension force is modeled using the Continuum Surface Force (CSF) approach of Brackbill, Kothe, and Zemach.\(^{30}\)

A numerical solution of the above system is obtained by using the open source fluid dynamics library OpenFOAM.\(^{31}\) The geometric VOF method of Roenby, Bredmose, and Jasak\(^{32}\) is used for the solution of Eq. 1, while Eq. 2 is solved using the PISO (Pressure-Implicit with Splitting of Operators) method for pressure-velocity coupling. The temporal and spatial derivatives in Eq. 2 are approximated using second-order schemes. For numerical stability, the convective term is discretized by the non-linear flux limiter of Jasak, Weller, and Gosman.\(^{33}\) Similar strategy has been used extensively in previous studies by the group e.g. in heat transfer\(^{34,35}\) and chemically reactive flow\(^{36}\) applications. Gradient terms at the cell faces are calculated with a second order central scheme with an explicit correction for mesh non-orthogonality.\(^{37}\) Solution of the pressure equation is obtained via a multigrid solver with Gauss-Seidel smoother. For time integration the second-order implicit backward scheme is used. The solver is validated against rising bubble benchmark problem of Hysing et al.\(^{38}\) in Appendix A.

B. Nozzle geometry

The studied large-scale asymmetric swirl nozzle is shown in Fig. 1. The flow enters the nozzle through an inlet pipe (I) which is aligned \( 45^\circ \) relative to the swirl chamber axis. The enlargement region (II) connects the pipe to the swirl chamber (III) through a single inlet port. As the entry is tangential to the chamber wall, the flow is forced into a swirling motion.
The rotating flow exits the nozzle via the discharge orifice (IV) located at the bottom of the swirl chamber. The orifice has a diameter of $d_o = 22$ mm and it is slightly off-centered from the chamber axis. Fig. 2 shows the main features of the two-phase bubbly flow inside the current nozzle design.
FIG. 3. Details of the computational domain and mesh.

C. Computational mesh

The used computational domain and mesh are shown in Fig. 3. Here, the same meshing strategy is used as in our previous work\textsuperscript{27,28}. We utilize an unstructured mesh consisting of a Cartesian background grid together with a body-fitted boundary layer mesh. Uniform cell size of $\Delta x = 0.3$ mm is used in the hexahedral part inside the nozzle and in the refinement region outside the nozzle. The outside refinement region extends 4.0 cm from the orifice. The selected resolution is based on our previous work and on the mesh sensitivity study of Appendix B. For single-phase inlet flow at the current parameter range, $\Delta x = 0.4$ mm has been previously found to be adequate for mesh insensitive solution of the inner-nozzle velocity fields and the developing air core at the orifice vicinity\textsuperscript{28}. In the present cases, new resolution requirements are imposed by the solution of the gas-liquid interfaces inside the nozzle. With the used resolution, the smallest (5 mm) bubbles introduced at the inlet are resolved with approximately 17 cells per diameter.

D. Boundary conditions

The resolved bubbly inlet condition is implemented with a uniform Dirichlet condition for the velocity ($u = U_b$, where $U_b$ is the mean bulk velocity) and a mapping procedure for
FIG. 4. Illustration of the bubbly inlet conditions inside the reference volume (cylinder). The instantaneous $\alpha$-field at the inlet plane is mapped from inside the reference volume at a cross-section that is marched through the volume in time.

The time-varying indicator field. For the pressure, a Neumann condition (zero gradient) is used at the inlet.

A cylindrical reference volume is used in implementing the mapped inlet condition for the indicator field as shown in Fig. 4. The mapping is carried out between a plane in the reference volume and the inlet plane of the nozzle. First, a volumetric field with the predefined bubble locations and properties is generated inside the reference volume. Because of the uniform velocity condition at the inlet, the axial location of the mapping plane inside the reference cylinder ($z$) is directly connected to the simulation time ($t = z/U_b$) and the plane can be marched through the cylinder in time. For the presently used long simulation times, the 40 cm long reference volume is periodically repeated. Temporary uniform Cartesian mesh with $\Delta x = 0.2$ mm is used for the mapping. Because the velocity at the inlet plane is uniform and thus the slip ratio is $S = u_g/u_l = 1$, the flow rate ratio $\beta$ is equivalent to the volume-averaged void fraction inside the reference cylinder. Therefore, the number of bubbles inside the reference volume is determined such that the volume-averaged void fraction is equal to $\beta$. The details of the bubbly inlet conditions in the different simulation cases are given in Sec. II E.
At the outlet boundary, a mixed boundary condition is applied for the variables depending on the flow direction. If the flow direction is towards the domain, Dirichlet conditions are used for both the velocity ($u$ extrapolated from inside the domain) and the indicator field ($\alpha = 0$). If the flow direction is outwards from the domain, zero gradient Neumann conditions are used for both fields. In both cases, a Dirichlet condition ($p = 0$ Pa) is used for the total pressure. At the walls, no-slip condition is imposed for the velocity, while the pressure and indicator fields have Neumann conditions.

E. Cases and parameters

In this paper, flow rate ratio and bubble diameter sweeps are carried out. The main flow parameters of the studied cases are shown in Table I, while the corresponding bubbly inlet conditions are illustrated in Fig. 4. To characterize the flows, we use the gas-to-total volumetric flow rate ratio, $\beta = Q_g/(Q_l + Q_g)^{39,40}$. In the table, the total mass flow rate ($\dot{m}$), the total volumetric flow rate ($Q = Q_l + Q_g$), and $\beta$ are related by $\dot{m} = [\beta \rho_g + (1 - \beta) \rho_l]Q$. The Reynolds number, $Re = \rho_l U_b d_p/\mu_l$, is referred to the liquid properties, the mean bulk velocity, and the inlet pipe diameter ($d_p$).

In the flow rate ratio sweep (cases V1–V3), the total mass flow is kept constant, while $\beta$ is varied ($\beta = 0.07$, 0.20 and 0.33). Hence, the total volumetric flow rate and mean velocity increase with increasing $\beta$. Furthermore, the bubbles are monodispersed and have a diameter of $d = 5$ mm. The bubbles are spatially randomly distributed inside the reference cylinder and do not touch the walls. At $\beta = 0.33$, overlap between the bubbles is allowed because of the difficulty of random packing at such high void fraction. In other cases, the bubbles do not initially touch each other.

The objective of the bubble diameter sweep (cases D1–D4) is to investigate the sensitivity of the downstream flow character to the exact location and size of the gas structures at inlet. For these cases, all flow parameters are kept constant, while the bubble diameter at the inlet is changed ($d = 5$, 10 and 15 mm). For simplicity, spherical shape is assumed for the bubbles. In addition to the three monodispersed cases, a polydispersed case with a distribution of bubble sizes between 5–15 mm is included. In the polydispersed case, the bubbles are divided into six size groups ($d \in \{5,7,9,11,13,15\}$ mm) between which the total gas volume is uniformly distributed.
TABLE I. Studied cases. The flow cases consist of the flow rate ratio (V1–V3) and bubble diameter (D1–D4) sweeps and the reference cases with single-phase inlet flow (S1–S4). Below, \( d \) refers to the initial bubble diameter.

| Case | \( \beta \) [-] | \( m \) [kg/s] | \( Q \) [L/s] | \( U_b \) [m/s] | \( Re \) [-] | GLR [-] | \( d \) [mm] |
|------|----------------|--------------|-------------|--------------|-------------|---------|-----------|
| V1   | 0.07           | 2.46         | 2.15        | 3.76         | 650         | 0.6\times 10^{-4} | 5          |
| V2   | 0.20           | 2.46         | 2.50        | 4.37         | 755         | 2.0\times 10^{-4} | 5          |
| V3   | 0.33           | 2.46         | 3.00        | 5.24         | 906         | 4.1\times 10^{-4} | 5          |
| D1   | 0.20           | 2.46         | 2.50        | 4.37         | 755         | 2.0\times 10^{-4} | 5          |
| D2   | 0.20           | 2.46         | 2.50        | 4.37         | 755         | 2.0\times 10^{-4} | 10         |
| D3   | 0.20           | 2.46         | 2.50        | 4.37         | 755         | 2.0\times 10^{-4} | 15         |
| D4   | 0.20           | 2.46         | 2.50        | 4.37         | 755         | 2.0\times 10^{-4} | 5–15       |
| S1   | 0              | 2.46         | 2.00        | 3.49         | 604         | 0       | -         |
| S2   | 0              | 2.64         | 2.15        | 3.76         | 650         | 0       | -         |
| S3   | 0              | 3.08         | 2.50        | 4.37         | 755         | 0       | -         |
| S4   | 0              | 3.69         | 3.00        | 5.24         | 906         | 0       | -         |

The particular choice of the mass/volumetric flow rates of the current cases with two-phase inlet flow is based on the previous cases of Laurila et al.\textsuperscript{28}. The previous work provides experimental references with single-phase inlet flow for the current setup. In addition to the experimental data, we repeat the simulations of the previous work with inlet conditions consistent with the present approach of uniform inlet velocity (cases S1–S4). In this paper, we use the term single-phase to refer to the cases S1–S4 with single-phase inlet flow.

In addition to the flow parameters given in Table I, the other presently used parameters are also based on the previous experiments/simulations of Laurila et al.\textsuperscript{28}. The liquid/gas densities and viscosities are \( \rho_l = 1230 \text{ kg/m}^3 \), \( \rho_g = 1 \text{ kg/m}^3 \), \( \mu_l = 192 \text{ mPas} \) and \( \mu_g = 0.0148 \text{ mPas} \), respectively, while the surface tension is \( \sigma = 0.064 \text{ N/m} \).
III. RESULTS AND DISCUSSION

A. Inlet pipe

First, bubble dynamics is investigated inside the inlet pipe. A developing two-phase flow is observed inside the inlet pipe as is illustrated by the instantaneous snapshots of the phase interface ($\alpha = 0.5$ iso-surface) shown in Fig. 5. As imposed by the inlet conditions, the gas phase initially consists of spherical bubbles which start to rapidly deform in the flow field. In the vicinity of the wall, the developing viscous boundary layer stretches and breaks up the bubbles (I in Fig. 5). In addition, the bubbles are observed to migrate towards the center of the pipe thus forming a nearly gas free wall layer. Such behavior is consistent with previous research on downflow in a vertical channel\(^{41-43}\). Near the pipe center, the bubbles start to deform and agglomerate. The agglomeration is affected by wake effects between the bubbles. Typically, the trailing bubble is entrained and merged in the wake of the preceding bubble (II). The merging process is similar to a collision between two rising bubbles\(^{44,45}\). In such a process, customarily the trailing bubble undergoes more deformation than the leading one\(^{46}\). The agglomeration leads to formation of large gas structures which exhibit slug-like behavior (III).

Bubble evolution in the near wall shear layer can be compared against literature on neutrally buoyant drops in shear flows (buoyancy neglected). Such flows are governed by the Reynolds number, the capillary number and the viscosity ratio\(^{47}\). Breakup is expected at intermediate bubble Reynolds numbers $Re_d \geq \mathcal{O}(10)^{48}$. If the Reynolds number is very low, e.g. for small bubbles, highly elongated stable shapes are possible at high capillary numbers ($Ca > 1$) and low viscosity ratios ($\lambda = \mu_g / \mu_l \ll 1$)\(^{49}\). Such elongated bubbles are not stable, if the capillary number exceeds the critical capillary number $Ca_c^{50,51}$. In the present cases, estimates for the non-dimensional numbers indicate that the initial bubbles are unstable due to high $Re_d = \rho \varphi \gamma a d / \mu_l \approx 45-470$ and high $Ca = \mu_l \dot{\gamma} a / \sigma \approx 4.2-14.6$. In all cases, $\lambda \approx 10^{-4}$. Above, $a = d/2$ is the bubble radius and $\dot{\gamma} = 4U_b / d_p$ is the shear rate assuming linear velocity profile inside the inlet pipe (diameter $d_p$).

The time evolution of the volume-averaged void fraction and total resolved interfacial area inside the inlet pipe are shown in Fig. 6. Here, the simulations with two-phase inlet flow are initialized from a flow field where the nozzle is filled with liquid. Hence, there is
FIG. 5. Instantaneous bubble fields inside the inlet pipe in (a) the flow rate ratio and (b) the bubble diameter sweeps. (I) Bubble stretching and break-up is dominant in the boundary layer near the walls of the pipe, while (II) in the center of the pipe bubbles agglomerate and form (III) larger, slug-like structures especially at higher $\beta$. The gravity is oriented in a $45^\circ$ angle relative to the pipe axis (see Fig. 3).

an initial transition period ($t \lesssim 0.05$ s) needed for the gas to occupy the entire length of the inlet pipe, i.e. the flow through time. This is evident from the linear rise at the start of the time series. After the initial transient the void fraction and interfacial area fluctuate around a statistical steady state. While the void fractions in the bubble diameter sweep are close to each other (as expected due to the same $\beta$), the computed interfacial areas differ significantly.

Axial profiles of the cross-section and time-averaged void fraction $\langle \alpha_g \rangle$ and slip ratio $S$ are shown in Fig. 7. The quantities are related by $\langle \alpha_g \rangle = Q_g / (SQ_l + Q_g)^{10}$. Here, the volumetric flow rates are constant along the pipe axis ($\dot{z}_p$) due to continuity. On the other hand, the void fraction diminishes due to the acceleration of the gas relative to the liquid
FIG. 6. Time series in the inlet pipe. (a) The volume-averaged void fraction and (b) the total resolved interfacial area fluctuate around the statistical steady state after the initial transient.

FIG. 7. Axial profiles in the inlet pipe. (a) The void fraction $\langle \alpha_g \rangle$ and (b) the slip ratio $S$ indicate that the two-phase flow is spatially developing in the pipe and at the entrance to the enlargement region.

(smaller cross-section needed for same volumetric flow at higher velocity). Correspondingly, the relative velocity difference between the phases increases along the pipe in all cases as is indicated by the increasing slip ratio $S = u_g/u_l^{40}$. This is also consistent with the observation of gas bubbles migrating towards the center of the pipe where flow velocities are higher. The results indicate that the two-phase pipe flow is still spatially developing in the inlet pipe and at the entrance to the enlargement region ($z_p \approx 24$ cm) in all the studied cases.

The probability density functions (PDFs) of bubble volume in the inlet pipe are shown in Fig. 8 as functions of an effective bubble diameter $d_i$ of size class $i$. Here, such PDFs are used
FIG. 8. Probability density functions of bubble volume in the inlet pipe. The PDFs in (a) the flow rate ratio and (b) the bubble diameter sweep indicate that most of the gas volume is included in the separate bubbles (discrete peaks) in the lower end of the effective bubble diameter ($d_i$) range.

to quantify the probability of finding gas volume in gas structures of size $d_i$. More precisely, time-averaged PDFs are calculated based on the instantaneous bubble counts $N_{i,j}$ collected at time instances $j$. As the total gas volume in a size class $i$ at time $j$ is $V_{i,j} \propto N_{i,j}d_i^3$, the normalized probability density function is calculated as:

$$\text{PDF}_i = \frac{\sum_j N_{i,j}d_i^3}{\Delta d \sum_k \sum_j N_{k,j}d_k^3}, \quad (3)$$

where $\Delta d = d_i - d_{i-1}$ is the width of the diameter size class which is constant for all classes.

The PDFs in Fig. 8 (a) show how the majority of the volume is in separate bubbles at $\beta = 0.07$ and 0.20 as illustrated by the discrete peaks at $d_i \lesssim 10$ mm. The multiple peaks are generated by bubble coalescence of two or more bubbles. Only at $\beta = 0.33$, large connected gas structures are present (e.g. peak at $d_i \approx 38$ mm) due to fast merging of the closely positioned bubbles and the allowed initial bubble overlap. Similarly in Fig. 8 (b), the PDFs are dominated by peaks from the initial 5, 10 and 15 mm bubbles and their combinations. It is noted, that although smaller structures are generated by shearing, they do not contain significant portion of the gas volume. The bubble volume PDFs highlight the large differences in the bubble fields inside the inlet pipe.
B. Enlargement region and swirl chamber

The enlargement region redirects the flow from the inlet pipe and forces the swirling motion inside the swirl chamber. Mean velocity fields together with exemplary instantaneous phase interface iso-surfaces are shown in Fig. 9. In the enlargement, two distinct regions are identified. In the high velocity region (I), the flow attaches near the bottom of the

![Diagram showing mean velocity magnitude and instantaneous bubble fields inside the enlargement region and swirl chamber.](image)

FIG. 9. Mean velocity magnitude (left) and instantaneous bubble fields (right) inside the enlargement region and swirl chamber. In the enlargement, the vertical extent of (I) the high velocity region increases with $\beta$. Gas is occasionally trapped in (II) the low velocity region. The dashed line and crosses mark the locations of the velocity profiles of Fig. 10. The cutplanes align with the pipe axis and the vertical midplane of the chamber.
FIG. 10. Mean velocity profiles inside the swirl chamber at 10–30 mm from the chamber roof (see Fig. 9 for profile locations). The two-phase ($\beta > 0$) profiles are compared to the current simulations and previous experiments$^{28}$ with single-phase inlet flow ($\beta = 0$) at corresponding mass and volumetric flow rates. At $\beta = 0.07$, the two-phase profiles are relatively close to the single-phase ones, while the differences become more pronounced as $\beta$ is increased.

enlargement, whereas there is low velocity region (II) in the upper part. Gas may be temporarily trapped in the low velocity region as illustrated in Fig. 9 (e.g. $\beta = 0.20$), while bubbles are relatively quickly convected along the high velocity stream. Similar separation between high and low velocity regions is observed both with single ($\beta = 0$) and two-phase ($\beta > 0$) inlet flows. In the two-phase cases, the extent of the high velocity region in the vertical direction increases with $\beta$ which includes both the effects of the bubbles and the increasing $Re$. The dynamics in the enlargement are affected by the Dean vortices$^{52,53}$ induced by the bend in the inlet pipe. The secondary flow structure consist of two counter-rotating Dean vortices in the plane perpendicular to the main flow direction.

The flow inside the swirl chamber is dominated by a strong central vortex. The vortex
FIG. 11. Probability density functions of bubble volume in the swirl chamber. (a) In the flow rate ratio sweep, bimodal distributions are observed in all cases with significant gas volume contained in the small bubbles. (b) In the bubble diameter sweep, the PDFs indicate similarity of the flows.

is visible from the tangential mean velocity profiles shown in Fig. 10. The plots show the two-phase ($\beta > 0$) velocity profiles together with the single-phase ($\beta = 0$) references of corresponding mass and volumetric flows at different distances from the chamber roof. In addition, previous experimental results are shown when available. Overall, the mean velocity profiles inside the chamber are similar in both the single and two-phase cases. This can be attributed to the fact that the same total volumetric flow rate is applied in both cases. Upon closer inspection of the profiles, it is noted that the local deviations from the single-phase references become more pronounced with increasing $\beta$. For example at $\beta = 0.33$ (30 mm), larger deviations at $y > 0$ can be observed due to the higher local presence of gas. However, it can be concluded that the main velocity characteristics inside the chamber are only moderately sensitive to $\beta$.

The bubble volume PDFs in the swirl chamber are depicted in Fig. 11. The PDFs are calculated with the same procedure as described in Sec. III A. In all cases, the distributions are bimodal. Contrary to the inlet pipe (see Fig. 8), a significant portion of the gas volume in the swirl chamber is contained in bubbles with small effective diameter (first peak at $d_i \lesssim 5$ mm). The second peak at higher $d_i$ consists of the larger gas structures which are more prevalent near the center of the vortex and, in practice, make up the gaseous core. Qualitatively, the gaseous core is intermittent, not necessarily continuous and consists of bubbles convecting from the enlargement region to the center of the vortex. The bubble
migration towards the center of rotation is consistent with previous literature\textsuperscript{23,24} and caused by the density difference and centrifugal acceleration. In Fig. 11 (a), there is a correlation between \( \beta \) and the size of the largest gas structures. The second, higher peak, which corresponds to the large structures, moves towards higher \( d_i \) as \( \beta \) is increased. In addition, the relative portion between small and large structures shifts towards the large structures with increasing \( \beta \). In contrast, the PDFs in Fig. 11 (b) show very close correspondence to each other. Interestingly, the inlet bubble diameter (5–15 mm) does not have a significant impact on the PDF. Consistent with this observation, also the polydispersed case behaves similarly.

C. Discharge orifice

Time-averaged velocity and void fraction profiles at the discharge orifice are shown in Fig. 12. The axial velocity profiles in Fig. 12 (a) are highly influenced by \( \beta \) as increasing velocities are observed in the orifice center. Opposite behavior is noted in the single-phase (\( \beta = 0 \)) flows with corresponding volumetric flow rates as the axial velocity decreases near the center of the vortex. This difference suggest that the cause of the increased axial velocity is indeed the increasing gas presence as opposed to the increased mean flow velocity. Again, the cases with different bubble size at \( \beta = 0.20 \) behave similarly. Contrary to the axial profiles, the tangential velocities in Fig. 12 (b) are less affected by \( \beta \). All cases exhibit qualitatively similar behavior, although a trend of increasing velocity with increasing \( \beta \) can be discerned.

Fig. 12 (c) depicts the time-averaged void fraction at the orifice. Although the peak values of the void fraction (\( \sim 0.5\text{-}0.7 \)) near the center of the orifice show no definite trend, the widths of the profiles increase with increasing \( \beta \). This indicates that at \( \beta = 0.07 \) gas is mainly present near the center of the orifice, while at higher \( \beta \) gas structures also occupy areas closer to the walls. Fig. 12 (d) confirms this observation also to other directions than the one where the profiles are taken. The effective diameter of the mean gaseous core as marked by the \( \alpha_g = 0.25 \) iso-contour increases with increasing gas flow. The iso-contours also illustrate the asymmetry of the flow at the orifice cross-section influenced by the nozzle geometry.

To further quantify the effective gaseous core diameter (\( d_{eff} \)), and therefore, the film
FIG. 12. Profiles at the discharge orifice. (a) Axial mean velocity. At the orifice center, the velocity increases in the two-phase ($\beta > 0$) cases and decreases in the single-phase ($\beta = 0$) cases as volumetric flow rate is increased. The markers denote single-phase cases with corresponding volumetric flow rates to $\beta = 0.07$ ($\triangledown$) and 0.33 ($\Diamond$). (b) Tangential mean velocity and (c) mean void fraction. (d) Iso-contours ($\alpha_g = 0.25$) at the orifice cross-section. The magenta dashed line illustrates the location of the profiles (a-c) and the arrows indicate the side of the inlet flow to the swirl chamber and the direction of rotation.

thickness at the orifice, time series of gas occupied orifice cross-sectional area based on the instantaneous $\alpha_g$ field is analyzed. At $\beta > 0$, gas is always present at the orifice and determination of the core size is not straightforward. To quantify the uncertainty, lower and upper bounds are estimated for $d_{eff}$. For the lower bound, the core diameter is determined from the cross-sectional area of the largest continuous gas patch, whereas the upper bound is based on the total cross-sectional area occupied by the gas.

Fig. 13 (a) shows how the effective diameter increases nearly linearly with volumetric flow
FIG. 13. (a) Effective gaseous core diameter ($d_{eff}$) increases with volumetric flow rate ($Q$). For $d_{eff}$ estimates in the two-phase ($\beta > 0$) cases, the lower and upper bounds are shown. (b) The increasing core existence factor ($F_{ave}$) indicates a developing core in the single-phase ($\beta = 0$) cases. The present results are compared to the results of Laurila et al.\textsuperscript{28}. The low existence factor partly explains the deviation in $d_{eff}$ at low $Q$.

In the two-phase cases ($\beta > 0$), the increase in the total flow rate $Q$ is due to increase in $\beta$ as the quantities are related by $Q = Q_l/(1 - \beta)$ where $Q_l$ is approximately constant. The effective diameter is estimated to be between 4.4–6.1, 5.9–7.5 and 7.5–8.9 for $\beta = 0.07$, 0.20 and 0.33, respectively. In addition, $d_{eff}$ in the single-phase ($\beta = 0$) cases is shown in the figure together with previous results of Laurila et al.\textsuperscript{28} (same flow parameters except for the parabolic velocity inlet condition). As expected, the inclusion of gas to the inlet flow increases the core diameter relative to the cases with single-phase inlet flow. However, the core size in the two-phase cases does not increase significantly faster as a function of $Q$ than in the single-phase cases. This may be partly attributed to the increasing axial velocity at orifice center where predominantly gas is present (see Fig. 12 (a) and (c)), i.e. the increased gas volumetric flow can be accounted for either by an increase in the gas velocity or gas cross-sectional area. In the present case, the results suggest the former.

The gaseous core existence factor, $F_{ave}$, is shown in Fig. 13 (b). Here, $F_{ave} \in [0,1]$ is defined as the fraction of time when a gaseous core of any size is present at the orifice plane. $F_{ave}$ is unity in the two-phase cases since the plane is, in practice, always occupied by some gas structure. In cases with single-phase inlet flow, the increasing core existence factor indicates a core in a developing state. The observed values are in good agreement.
with previous results\textsuperscript{28}. In the single-phase cases, the larger variability of $d_{eff}$ at low $Q$ is partly attributed to the low existence factor which decreases the effective averaging time.

D. Liquid film

The effect of the gas presence on the ejected liquid film is highly important with regard to subsequent droplet formation and spray characteristics. As illustrated in Fig. 14, the film remains mainly continuous in the near-nozzle region (4 cm from the orifice). However, the films become more corrugated with increasing $\beta$ in comparison to the smooth surface in the single-phase case. The added gas increases the irregularity of the film especially in the center of the conical spray. This is in line with the observation that the larger gas bubbles tend to migrate near the center of the swirl chamber, and therefore, affect the gas core vicinity and inward facing part of the film the most. In addition, the cross-sections reveal small bubbles

![Instantaneous snapshots of the gas-liquid surface](image)

**FIG. 14.** Instantaneous snapshots of the gas-liquid surface illustrate the increasing number of surface features as $\beta$ is increased. At bottom row, the cut view reveals a growing gas core and small bubbles entrapped inside the liquid film.
FIG. 15. Total resolved interfacial area in the spray region as a function of the total volumetric flow rate $Q$. The area in the two-phase cases (2-ph.) is higher than in the corresponding single-phase cases (1-ph.). Contrary to $Q$ and $\beta$, the initial bubble size at the inlet has only a moderate effect on the spray area.

Substantial droplet formation is not observed in the near-nozzle region (0–4 cm). For this reason, we utilize the total resolved interfacial area to quantify the film’s susceptibility to breakup. Large interfacial area indicates either high film roughness or large number of in-film bubbles, both of which promote breakup. The interfacial area in the spray region is shown in Fig. 15. First, it is noted that the inclusion of gas bubbles in the inlet stream clearly increases the surface area compared to the case with single-phase inlet flow at the corresponding volumetric flow rate. In fact, there is a linear relationship between $Q$ and the area. Moreover, relative increases of approximately 6, 20 and 40% are observed at $\beta = 0.07, 0.20$ and 0.33, respectively. It should be noted, that in the case of two-phase inlet flow, the increase in the flow rate $Q$ is due to increase in $\beta$ as mentioned above. On the other hand, changes in the inlet bubble diameter have only a moderate effect on the area which is consistent with the observation of minor differences with regard to orifice and swirl chamber quantities. There is an opposite, decreasing trend in the single-phase ($\beta = 0$) cases. This is attributed to the film being in a transition from an S-shaped to conical film type at the present range, i.e. the film is straightening and thus the area is reduced.
FIG. 16. Time-averaged indicator field at a plane 4 cm below the orifice. Spray uniformity increases with $\beta$. Orifice center is marked with a dot.

The flow rate ratio also affects the uniformity of the spray. Here, the time-averaged indicator field shown in Fig. 16 is used to illustrate the uniformity of the near-nozzle spray at a plane 4 cm below the orifice. Consistent with previous results\textsuperscript{28}, the sprays are tilted and asymmetric in the present parameter range due to the influence of the asymmetric nozzle geometry. For both the bubble-free ($\beta = 0$) and bubbly ($\beta > 0$) cases, the spray uniformity is improved with volumetric flow rate. It is worth noting that the uniformity is significantly enhanced at $\beta = 0.33$ due to bubble inclusion at the inlet.

IV. CONCLUSIONS

Bubbly flow in a large-scale swirl atomizer was studied at a parameter range relevant for injection of high-viscosity fuels. Gas in the form of large bubbles (5–15 mm) was introduced at the inlet to study the effect of the two-phase flow on the nozzle operation. Gas-to-total volumetric flow rate ratio sweep ($\beta = 0.07$, 0.20 and 0.33) and bubble diameter sweep at $\beta = 0.20$ were carried out to study the nozzle performance and sensitivity to $\beta$. Also, the results were linked to our previous findings at $\beta = 0$. The computational setup provides insight relevant for injection scenarios where significant amount of gas is present inside the nozzle e.g. due to liquid aeration or flash boiling. The main findings of the paper can be summarized as:

1. It was observed that $\beta$ had a profound effect on the inner-nozzle flow. The bubble
volume PDFs in the swirl chamber were noted to be bimodal and strongly affected by $\beta$. Both the size of the largest gas structures and the scale-separation between the small and large bubbles increased with $\beta$. At the discharge orifice ($d_o$), the effective gas core diameter ($d_{eff}$) increased with $\beta$ and was found to be in the range $0.2 < d_{eff}/d_o < 0.4$. In addition, the core was always present at $\beta > 0$, whereas at $\beta = 0$ the core was intermittent.

2. It was found that $\beta$ significantly affected the liquid film outside the nozzle. The effect was quantified by the increase in the resolved interfacial area of the film. The area was noted to increase by 6, 20 and 40% relative to the single-phase flow ($\beta = 0$) with the same volumetric flow rate at $\beta = 0.07$, 0.20 and 0.33, respectively. In addition, the spray uniformity was clearly enhanced at $\beta = 0.33$ when compared to the corresponding single-phase case.

3. For the representative case of $\beta = 0.20$, the exact bubble size at the inlet had only a minor effect on the flow dynamics. The downstream flow character was found to be insensitive to the initial bubble diameter (5–15 mm) as noted in particular from the bubble volume PDFs in the swirl chamber.

In summary, we have provided numerical evidence that bubble inclusion at the inlet affects global liquid film characteristics with relevance to atomization. We highlight the rich physics inside effervescent-swirl nozzles including bubble coalescence, bubble accumulation inside the main vortex, and faster destabilization of the liquid film with increasing $\beta$.

ACKNOWLEDGMENTS

The authors acknowledge ANDRITZ Oy. for financial support. The computational resources for this study were provided by CSC - Finnish IT Center for Science and the Aalto Science-IT project.

DATA AVAILABILITY

The data that support the findings of this study are available from the corresponding author upon reasonable request. Restrictions apply to the availability of the nozzle geom-
etry, which was used under license for this study. Data is available from the authors upon reasonable request and with the permission of ANDRITZ Oy.

Appendix A: Validation

The numerical method is validated against the widely used\textsuperscript{54–57} benchmark cases of Hysing\textit{ et al.}\textsuperscript{38}. The tests consist of two rising bubble cases with different surface tension, density and viscosity ratios. The main flow parameters are given in Table II, while more accurate description of the setup can be found in Hysing\textit{ et al.}\textsuperscript{38}. Here, the grid sensitivity of the solution is investigated at the interval $20 \leq D/\Delta x \leq 160$, where $D$ is the initial diameter of the bubble. The numerical schemes employed in the benchmark cases are same as in the final simulations.

Figs. 17 and 18 show the time evolution of the center of mass and rise velocity of the bubble, and its final shape at $t = 3$ s for cases 1 and 2, respectively. The results are compared to the validation data of Hysing\textit{ et al.}\textsuperscript{38} (TP2D solver, $D/\Delta x = 160$) and Klostermann, Schaake, and Schwarze\textsuperscript{54} (OpenFOAM, $D/\Delta x = 160$). The current results are in good agreement with Klostermann, Schaake, and Schwarze\textsuperscript{54} and other previous OpenFOAM

| Case | $\rho_1$ [kg/m$^3$] | $\rho_2$ [kg/m$^3$] | $\mu_1$ [Pas] | $\mu_2$ [Pas] | $g$ [m/s$^2$] | $\sigma$ [N/m] |
|------|---------------------|---------------------|---------------|---------------|---------------|---------------|
| 1    | 1000                | 100                 | 10            | 1             | 0.98          | 24.5          |
| 2    | 1000                | 1                   | 10            | 0.1           | 0.98          | 1.96          |

**FIG. 17.** Benchmark case 1. Time evolution of (a) the center of mass and (b) the rise velocity of the bubble, and (c) its final shape at $t = 3$ s.
FIG. 18. Benchmark case 2. Time evolution of (a) the center of mass and (b) the rise velocity of the bubble, and (c) its final shape at $t = 3 \text{s}$.

studies\textsuperscript{57}. However, expected deviations are observed with respect to the data of Hysing et al.\textsuperscript{38}. In case 1, already the coarsest mesh captures the bubble evolution, while in case 2, convergence towards the reference is observed with mesh refinement. In case 2, the mesh convergence behavior is similar to previously published results\textsuperscript{54,57}. Overall, good correspondence with the reference solution is demonstrated.

Appendix B: Mesh sensitivity

The studied cases pose complex transient two-phase flow developing throughout the nozzle. For single-phase inlet flows, the present flow solution is considered as highly-resolved. However, the smallest bubbles are not fully resolved and the results could be in practice subject to implicit filtering effects of the finite mesh resolution. Next, we investigate the sensitivity of the numerical solution to the mesh resolution. Here, one of the cases ($\beta = 0.20$, $d = 5 \text{ mm}$) is computed on four grids (M1–M4) and the base cell size of the hexagonal background grid is varied. The cell sizes and the total number of cells of the meshes are presented in Table III. The selected cell sizes are based on our previous work (single-phase inlet flow), where resolution similar to M3 was concluded to adequately resolve the developing air core in the orifice vicinity\textsuperscript{28}. In the mesh assessment, the time step is varied to maintain Courant number of $Co = 0.5$, while all other parameters are kept constant between the runs. For M1–M3, the initial conditions are interpolated from the densest mesh.

Mean velocity profiles inside the swirl chamber and at the discharge orifice are shown in Fig. 19 (a)-(c). It is noted, that the mean profiles are relatively well captured already.
TABLE III. The base cell size and the total number of cells in the meshes.

| Mesh | $\Delta x$ [mm] | $N_{\text{cells}}$ [million] |
|------|-----------------|-------------------------------|
| M1   | 1.00            | 0.79                          |
| M2   | 0.55            | 4.19                          |
| M3   | 0.40            | 10.97                         |
| M4   | 0.30            | 22.87                         |

FIG. 19. (a) Tangential mean velocity profiles inside the swirl chamber and (b) axial and (c) tangential profiles at the orifice show only moderate variation over a wide range of $\Delta x$. Inlet pipe axial profiles of (d) the void fraction and (e) the slip ratio converge for finer mesh resolutions. (f) Increasingly smaller bubbles are resolved with decreasing $\Delta x$ which weights the bubble volume PDFs towards low $d_i$.

with coarser meshes. At locations of high void fraction (e.g. at the orifice center or behind the inlet port at swirl chamber, $5 \leq y \leq 15$ mm), moderate deviations are observed in the results. In order to evaluate the effect on global quantities, Fig. 19 (d) and (e) show the axial profiles of the time and cross-section averaged void fraction and slip ratio. Convergence of the results is demonstrated as $\Delta x$ is decreased.

The bubble volume PDFs in the swirl chamber are shown in Fig. 19 (f). The relative
weight of the small diameter bubbles of the total gas volume is increased with increasing mesh resolution. In general, the cell size is directly connected to the minimum bubble size that can be resolved. Hence, the smallest resolved bubble is limited by $\Delta x$. We acknowledge that the smallest bubbles present in the current setup cannot be resolved with reasonable grid resolution.

Even though the mean velocity profiles are fairly insensitive to the resolution and M3 performs well regarding the pipe axial quantities, the finest M4 grid is employed in the final simulations. This choice aids in correct solution of the bubble dynamics of the smaller gas structures and allows to resolve the largest possible portion of the bubble spectrum within the limitations of the presently available computational resources.

REFERENCES

1. S. Sovani, P. Sojka, and A. Lefebvre, “Effervescent atomization,” Progress in Energy and Combustion Science 27, 483 – 521 (2001).
2. D. Konstantinov, R. Marsh, P. J. Bowen, and A. Crayford, “Effervescent atomization for industrial energy – technology review,” Atomization and Sprays 20, 525-552 (2010).
3. M. P. Järvinen, A. E. P. Kankkunen, R. Virtanen, P. H. Miikkulainen, and V. P. Heikkilä, “A one-dimensional two-phase flow model and experimental validation for a flashing viscous liquid in a splash plate nozzle,” Atomization and Sprays 25, 795 – 817 (2015).
4. P. Miikkulainen, A. Kankkunen, M. Järvinen, and C.-J. Fogelholm, “The significance of velocity in black liquor spraying,” TAPPI journal 8, 36–40 (2009).
5. A. Lefebvre and V. McDonell, Atomization and Sprays, second edition ed. (CRC Press, Boca Raton, 2017).
6. G. A. Vijay, N. S. V. Moorthi, and A. Manivannan, “Internal and external flow characteristics of swirl atomizers: A review,” Atomization and Sprays 25, 153–188 (2015).
7. Z. Kang, Z. Wang, Q. Li, and P. Cheng, “Review on pressure swirl injector in liquid rocket engine,” Acta Astronautica 145, 174 – 198 (2018).
8. M. Ochowiak, L. Broniarz-Press, and J. Rozanski, “The discharge coefficient of effervescent atomizers,” Experimental Thermal and Fluid Science 34, 1316 – 1323 (2010).
9. J. Jedelsky and M. Jicha, “Unsteadiness in effervescent sprays: a new evaluation method and the influence of operational conditions,” Atomization and Sprays 18, 49–83 (2008).
X. Huang, X. Wang, and G. Liao, “Visualization of two phase flow inside an effervescent atomizer,” Journal of Visualization 11, 299–308 (2008).

K.-C. Lin, P. Kennedy, and T. Jackson, “Structures of internal flow and the corresponding spray for aerated-liquid injectors,” in 37th Joint Propulsion Conference and Exhibit (2001) p. 3569.

D. Sen, M. A. Balzan, D. S. Nobes, and B. A. Fleck, “Bubble formation and flow instability in an effervescent atomizer,” Journal of Visualization 17, 113–122 (2014).

M. Hong, B. A. Fleck, and D. S. Nobes, “Unsteadiness of the internal flow in an effervescent atomizer nozzle,” Experiments in fluids 55, 1855 (2014).

C. Sun, Z. Ning, X. Qiao, M. Lv, J. Fu, J. Zhao, and X. Wang, “Numerical simulation of gas-liquid flow behavior in the nozzle exit region of an effervescent atomizer,” International Journal of Spray and Combustion Dynamics 11, 1756827718821592 (2019).

C. Sun, Z. Ning, X. Qiao, M. Lv, Y. Li, J. Zhao, and X. Wang, “Study on effervescent spray morphology based on internal gas-liquid two-phase flow patterns,” European Journal of Mechanics - B/Fluids 74, 123 – 138 (2019).

M. Mousavi and A. Dolatabadi, “Numerical study of the effect of gas-to-liquid ratio on the internal and external flows of effervescent atomizers,” Transactions of the Canadian Society for Mechanical Engineering 42, 444–456 (2018).

L. Pan, G. Yang, and B. Li, “Numerical investigation of two-phase flow in and effervescent atomizer for water mist fire suppression,” Atomization and Sprays 21, 819–831 (2011).

B. Li, L. Pan, and G. Yang, “Numerical studies of the flow structure in the final discharge orifice of effervescent atomizer,” Atomization and Sprays 22, 259–274 (2012).

Z. A. Kaklar and M. R. Ansari, “Numerical analysis of the internal flow and the mixing chamber length effects on the liquid film thickness exiting from the effervescent atomizer,” Journal of Thermal Analysis and Calorimetry 135, 1881–1890 (2019).

D. Law and T. G. Shepard, “Three-dimensional volume of fluid simulations of air bubble dynamics in a converging nozzle,” in Fluids Engineering Division Summer Meeting (2018).

S. A. Esfarjani and A. Dolatabadi, “A 3d simulation of two-phase flow in an effervescent atomizer for suspension plasma spray,” Surface and Coatings Technology 203, 2074 – 2080 (2009).

K. Pougatch, M. Salcudean, and J. McMillan, “Influence of mixture non-uniformity on the performance of an effervescent nozzle,” Fuel 116, 601 – 607 (2014).
23B. Maneshian, K. Javadi, and M. Taeibi Rahni, “Bubble dynamics in rotating flow under an accelerating field,” Physics of Fluids 30, 082108 (2018).

24E. A. Van Nierop, S. Luther, J. J. Bluemink, J. Magnaudet, A. Prosperetti, and D. Lohse, “Drag and lift forces on bubbles in a rotating flow,” Journal of Fluid Mechanics 571, 439–454 (2007).

25C.-T. Hsiao, J. Ma, and G. L. Chahine, “Numerical study of gravity effects on phase separation in a swirl chamber,” Chemical Engineering Science 165, 177 – 185 (2017).

26M. Ochowiak, “Discharge coefficient of effervescent atomizers with the swirl motion phenomenon,” Experimental Thermal and Fluid Science 79, 44 – 51 (2016).

27E. Laurila, J. Roenby, V. Maakala, P. Peltonen, H. Kahila, and V. Vuorinen, “Analysis of viscous fluid flow in a pressure-swirl atomizer using large-eddy simulation,” International Journal of Multiphase Flow 113, 371 – 388 (2019).

28E. Laurila, S. Koivisto, A. Kankkunen, K. Saari, V. Maakala, M. Järvinen, and V. Vuorinen, “Computational and experimental investigation of a swirl nozzle for viscous fluids,” International Journal of Multiphase Flow 128, 103278 (2020).

29C. Hirt and B. Nichols, “Volume of fluid (vof) method for the dynamics of free boundaries,” Journal of Computational Physics 39, 201 – 225 (1981).

30J. Brackbill, D. Kothe, and C. Zemach, “A continuum method for modeling surface tension,” Journal of Computational Physics 100, 335 – 354 (1992).

31H. G. Weller, G. Tabor, H. Jasak, and C. Fureby, “A tensorial approach to computational continuum mechanics using object-oriented techniques,” Computers in Physics 12, 620 – 631 (1998).

32J. Roenby, H. Bredmose, and H. Jasak, “A computational method for sharp interface advection,” Royal Society Open Science 3 (2016), 10.1098/rsos.160405.

33H. Jasak, H. Weller, and A. Gosman, “High resolution NVD differencing scheme for arbitrarily unstructured meshes,” International Journal for Numerical Methods in Fluids 31, 431 – 449 (1999).

34P. Peltonen, K. Saari, K. Kukko, V. Vuorinen, and J. Partanen, “Large-eddy simulation of local heat transfer in plate and pin fin heat exchangers confined in a pipe flow,” International Journal of Heat and Mass Transfer 134, 641 – 655 (2019).

35A. Laitinen, K. Saari, K. Kukko, P. Peltonen, E. Laurila, J. Partanen, and V. Vuorinen, “A computational fluid dynamics study by conjugate heat transfer in openfoam: A liquid
cooling concept for high power electronics,” International Journal of Heat and Fluid Flow 85, 108654 (2020).

36 H. Kahila, A. Wehrfritz, O. Kaario, and V. Vuorinen, “Large-eddy simulation of dual-fuel ignition: Diesel spray injection into a lean methane-air mixture,” Combustion and Flame 199, 131 – 151 (2019).

37 H. Jasak, Error analysis and estimation for the finite volume method with applications to fluid flows, Ph.D. thesis, Imperial College London (1996).

38 S. Hysing, S. Turek, D. Kuzmin, N. Parolini, E. Burman, S. Ganesan, and L. Tobiska, “Quantitative benchmark computations of two-dimensional bubble dynamics,” International Journal for Numerical Methods in Fluids 60, 1259–1288 (2009).

39 C. E. Brennen, Fundamentals of Multiphase Flow (Cambridge University Press, Cambridge, 2005).

40 G. Yadigaroglu and G. F. Hewitt, Introduction to Multiphase Flow: Basic Concepts, Applications and Modelling (Springer International Publishing, Cham, 2018).

41 J. Lu, S. Biswas, and G. Tryggvason, “A DNS study of laminar bubbly flows in a vertical channel,” International Journal of Multiphase Flow 32, 643 – 660 (2006).

42 J. Lu and G. Tryggvason, “Numerical study of turbulent bubbly downflows in a vertical channel,” Physics of Fluids 18, 103302 (2006).

43 X. Chu, Y. Liu, W. Wang, G. Yang, B. Weigand, and H. Nemati, “Turbulence, pseudo-turbulence, and local flow topology in dispersed bubbly flow,” Physics of Fluids 32, 083310 (2020).

44 S. Hosokawa and A. Tomiyama, “Effects of bubble wake on coalescence between planar bubbles,” Journal of Fluid Science and Technology 1, 94–104 (2006).

45 A. Hadidi and M. E. Nimvari, “A geometric mass control approach in level set method to simulate multiphase flows with complex interface topologies, case study: Oblique coalescence of gas bubbles in a liquid,” Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering 234, 56–69 (2020).

46 A. Zhang, Z. Guo, Q. Wang, and S. Xiong, “Three-dimensional numerical simulation of bubble rising in viscous liquids: A conservative phase-field lattice-boltzmann study,” Physics of Fluids 31, 063106 (2019).

47 A. Komrakova, O. Shardt, D. Eskin, and J. Derksen, “Lattice boltzmann simulations of drop deformation and breakup in shear flow,” International Journal of Multiphase Flow
59, 24 – 43 (2014).

48G. Ryskin and L. G. Leal, “Numerical solution of free-boundary problems in fluid mechanics. part 3. bubble deformation in an axisymmetric straining flow,” Journal of Fluid Mechanics 148, 37–43 (1984).

49H. A. Stone, “Dynamics of drop deformation and breakup in viscous fluids,” Annual Review of Fluid Mechanics 26, 65–102 (1994).

50X. Zhao, “Drop breakup in dilute newtonian emulsions in simple shear flow: New drop breakup mechanisms,” Journal of Rheology 51, 367–392 (2007).

51N. Barai and N. Mandal, “Breakup modes of fluid drops in confined shear flows,” Physics of Fluids 28, 073302 (2016).

52W. R. Dean, “Note on the motion of fluid in a curved pipe,” The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science 4, 208–223 (1927).

53S. A. Berger, L. Talbot, and L. S. Yao, “Flow in curved pipes,” Annual Review of Fluid Mechanics 15, 461–512 (1983).

54J. Klostermann, K. Schaake, and R. Schwarze, “Numerical simulation of a single rising bubble by VOF with surface compression,” International Journal for Numerical Methods in Fluids 71, 960–982 (2013).

55R. Subburaj, P. Khandelwal, and S. Vengadesan, “Numerical study of flow past an elliptic cylinder near a free surface,” Physics of Fluids 30, 103603 (2018).

56Y. Cao and R. Macián-Juan, “Numerical investigation of central breakup of large bubble induced by liquid jet,” Physics of Fluids 32, 033302 (2020).

57L. Gamet, M. Scala, J. Roenby, H. Scheufler, and J.-L. Pierson, “Validation of volume-of-fluid OpenFOAM® isoAdvector solvers using single bubble benchmarks,” Computers & Fluids 213, 104722 (2020).
\[ Q = Q_l / (1 - \beta) \]
\[ Q_l \approx \text{const.} \]

Interfacial area \( [\text{cm}^2] \)

\( Q \) [L/s]

2-ph. (5 mm)
2-ph. (10 mm)
2-ph. (15 mm)
2-ph. (5-15 mm)
1-ph.
