

Computational fluid dynamics modelling of air entrainment for a plunging jet

Fatemeh Salehi^{1,*}, Esmail Ajdehak², Yannis Hardalupas³

¹School of Engineering, Macquarie University, Sydney, Australia

²Jacobs Engineering, Sydney Australia

³Department of Mechanical Engineering, Imperial College London, London, UK

* Corresponding author (fatemeh.salehi@mq.edu.au)

Abstract

This study focuses on computational fluid dynamics (CFD) modelling of the air entrainment phenomena in water columns which are commonly used in processing plants. The CFD model is first validated against experimental data. Then, a comprehensive set of CFD simulations are conducted to understand the effect of inlet jet velocity, nozzle diameter, tank diameter, and tank height on the bubble formation. Three main patterns are identified: dispersed bubbles, medium air pockets (MAP) and large air pockets (LAP). For the dispersed bubble case, the initially formed air pockets break into smaller bubbles and hence the water level rises linearly. For the MAP case, the size of the air pockets remains larger than for the dispersed bubble case, though, with time they burst at the surface. Finally, for the LAP case, the inkling jet interacts violently with the wall and hence large air pockets are trapped within the column, causing the overflow of the fluid. The flows are further analysed using non-dimensionless Reynolds and Froude numbers, providing an acceptable range of operating parameters that ensure the fluid does not overflow.

Keywords: volume-of-fluid (VOF); air entrainment; impinging water jet; two-phase flows

1. Introduction

Air entrainment into water impinging on surfaces occurs in both natural and industrial processes. In a natural process, this process entrains oxygen and carbon dioxide in the water at the free surface of oceans and rivers, affecting the survival of ecosystems (Ramirez de la Torre et al. 2020). In a variety of industrial processes, air entrainment phenomena into liquids are also present (Brouilliot and Lubin 2013, Kulkarni and Patwardhan 2014, Okhovat-Alavian et al. 2019, Schulz and Bart 2019, Cervantes-Álvarez et al. 2020). In some applications, such complex two-phase flow benefits the industrial process, while in other applications it may be a disadvantage for the design of components. In chemical reactors (Bhusare et al. 2017, Guo et al. 2017), for example, aeration plays an important role to achieve the end products, while in mould-filling processes, air entrapment forms oxide-inclusion in the product that leads to additional repairs in ferrous and non-ferrous castings.

Such phenomenon is also relevant to water treatment and various processing plants that involve a complex high pressure piping system. A hydraulic break column (HBC) is introduced to break the pressure within a system, normally, to atmospheric pressure. To achieve this, the common practice is to run the pressure pipe into a top open HBC where the column is drained from the bottom section into a gravity system pipeline. On some occasions due to poor design, the HBCs are overflowed causing chemical contaminations to the surrounding environments. This is due to the air entrainment phenomenon occurring during the process, where the HBC rate of draining falls behind an air entrained filling height rate. The flow inside an HBC involves a complex transient two-phase flow process where an impinging jet on an empty bottom surface or a liquid surface of a partially filled HBC generates highly turbulent regimes. Hence, as a result, air bubbles/pockets are formed during the filling process. This causes the outflow velocity to deviate from the theoretical value that is linearly correlated to the inlet flow rate and hence the water level in the column rises and overflows.

The mechanism of air bubble formation for a plunging jet to a water pool has been studied experimentally in the literature where most studies focused on the average and global dynamic behaviour of the air/water mixture under the free liquid surface as the result of the plunging jet flow (Zhu et al. 2000, Kiger and Duncan 2012, Qu et al. 2013, Miwa et al. 2018, Ramirez de la Torre et al. 2020). The air entrainment phenomenon depends on the physical properties of the fluids, geometrical characteristics of the plunging jet system, and operational conditions such as fluid velocity (Metzger and Kind 2015, Wang et al. 2018). The main focal point of such studies was to establish boundaries to differentiate the air bubble formation regimes under the

free surface. As a result, four major regimes below the water pool free surface are identified to address the concentration of air inside the water: (1) no-entrainment, (2) incipient entrainment, (3) intermittent entrainment, and (4) continuous entrainment (Biń 1993, Chirichella et al. 2002, El Hammoumi et al. 2002). These regimes are characterised using the impact jet velocity, which is defined as

$$V_j = \sqrt{V_0^2 + 2gL_j} \quad (1)$$

where V_0 is the water velocity at the nozzle exit, g is the gravitational acceleration and L_j is the falling length, which is defined as the distance between the nozzle exit and water free surface. For small V_0 , no entrainment was observed while with further increasing the jet velocity, the flow regime moves to incipient and intermittent entrainment followed by continuous entrainment (Qu et al. 2013, Roy et al. 2013).

Previous studies on the bubble size and interfacial phenomena in different types of bubble column showed that the column's dimensions, gas velocity, and the level of turbulence affect the bubble size distribution. It was demonstrated that the initial bubble size distribution created at the bottom of the column is carried out throughout the height of the column. This is due to the equilibrium between the coalescence and break-up. However, the effect of break-up increases, as the gas velocity increases due to its major effect on the hydrodynamics and consequently on the bubble size distribution (Lau et al. 2013). In the bubbly flow zone, further away from the bottom of the column, bubble size are larger due to coalescence and kinetic energy distribution (Mandal et al. 2005, Majumder et al. 2006). The dispersion coefficient of bubble motion is also affected by the column diameter as well as the gas velocity which is due to effect of fluid circulation and interaction of different size of bubble. On the other hands, the velocity distribution depends on the turbulence of the phase (Majumder 2008).

The air entrainment phenomenon has been also studied using computational fluid dynamic (CFD) simulations (Torré et al. 2008, Karim et al. 2017). Such multiphase flows are highly transient processes with a complex temporal and spatial variation of the interface between the liquid and gaseous phases. It has been found that the air entrainment at surface discontinuities is associated with turbulent kinetic energy and jet velocity (Xiang et al. 2014, Majidi and Beckermann 2019). Simulations have been conducted using both Reynolds averaged Navier Stokes (RANS) and large eddy simulation (LES) methods. (Khezzar et al. 2015) applied a multiphase LES Volume-of-Fluid (VOF) model to simulate the flow of two turbulent plunging

water jets, achieving good agreement with the experimental data while topological phenomena taking place during the transient impact and continuous entrainment regime was well captured. (Lopes et al. 2016) also adopted a VOF method to study a 3D circular plunging jet. They developed an explicit term to detect bubble formation and air-entrainment at the free surface that improved the flow prediction in terms of bubble formation and free-surface wave's amplitude. Further study by (Zidouni Kendil et al. 2012) focused on the development of the flow field below the pool surface. In this study, the jet was modelled as a two-phase bubbly flow injecting into the pool that neglected the complex interactions between surface waves and turbulence that leads to the entrainment of air. Satisfactory predictions of mean flow quantities were achieved. It was found that in the wake of bubbles, the liquid is dragged towards the free surface. Although the lift coefficient had a marginal impact on the velocity field, the combination of different values for the initial bubble size, void fraction concentrations, and the lift coefficient affected the plume shape. (Ma et al. 2010) developed a subgrid model that assumes that at low liquid jet impact velocities air is entrained due to the roughness of the liquid jet, although at higher velocities this is due to the formation and breakdown of an air cavity near the impact point. They coupled the model with an Eulerian multi-fluid (MF) method to simulate air entrainment beneath a plunging liquid jet for a range of jet velocities and achieved good agreement with the measured data.

For a plunging jet on a water pool, the local surface oscillation is generated at the contact point when the jet vertically enters the pool. Eventually, the pool surface undulation is affected and air pockets are created underneath the pool surface. Consequently, the air is entrained through the vacant space generated by the pocket. The generated air pocket interface is highly unstable and hence the pocket disintegrates into multiple small bubbles. The entrained bubbles are then carried away within the pool by the large-scale eddies, and it has been found that most of these large-scale vortices are oriented orthogonal to the flow direction (Miwa et al. 2018).

Previous studies mainly focused on a plunging jet to a liquid pool, where the level of the water pool remains constant. This is different from the HBC process where the water level varies with time subject to how the draining rate through the downstream pipework operates. Although similar processes occur in the HBC process, the dominant cause of air entrainment is the formation of large air pockets. The filling process of an HBC is a combination of the water impingement jet (Dewan et al. 2012) and the plunging jet to a water pool that simultaneously lead to air entrainment. The HBC is initially empty. The vertical water jet impinges on the bottom surface of the column and then the flow direction changes, which also

causes a momentum force at the bottom of the tank, creating a water layer at the bottom surface of the column. This is a common design method of cooling systems in different applications (Yang et al. 2015). As the water flow along the bottom surface reaches the tank's sidewall, it moves vertically due to its momentum along the column's sidewall. The jet kinetic energy gradually converts to potential energy, and hence the water reaches a maximum height (fountain height) then travels down in a fountain profile path towards the centre of the tank before reaching to the bottom surface again. This causes air encapsulation and creates large-medium air pockets. Depending on the jet velocity and tank dimensions, the air pockets may break into smaller bobbles. Simultaneously, the water level in the column increases and hence the continuation of the water jet causes instabilities at the water level surface that also creates bubbles.

A reliable design of HBCs requires a better understanding of the air entrainment process to ensure cost-effective operation, while it prevents harmful environmental impacts of overflowing contaminants. Currently, there is a lack of data and procedures for such design. This study, for the first time, aims to use CFD two-fluid modelling to understand the correlation between air entrainment with operational and geometrical parameters of HBCs including inlet jet velocity, nozzle diameter, tank diameter, and tank height. This will provide improved design guidelines for industrial processes.

2. Problem definition

The schematic view of HBC is presented in Fig. 1. The HBC is initially empty. The high momentum water jet impinges the bottom surface of the tank and then it flows parallel to the surface until it reaches the sidewalls, where the water flows upwards. Depending on the initial momentum of the water (i.e. nozzle diameter and jet velocity) and the height and diameter of the tank, the water reaches the 'fountain' height h and then the 'fountain' breaks. To understand the air entrainment during the filling process of a tank, effects of main parameters, including inlet jet velocity, V_0 , nozzle diameter, d , tank diameter, D , and tank height, H , were considered. The range of values for each parameter are considered as 0.025, 0.05, and 0.08 m for nozzle diameter (d); 0.3 and 0.5 m for tank diameter (D); 1 and 2 m for tank height (H); and 1, 2, 3, and 4 m/s for inlet jet velocity, V_0 .

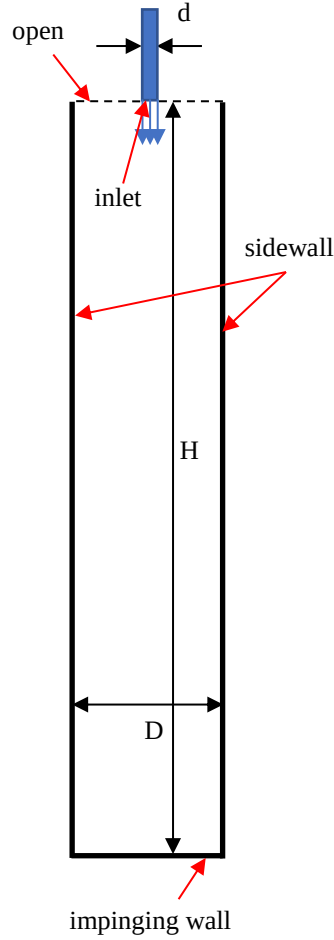


Figure 1: Schematic of the water hydraulic break column.

3. Methodology

This section briefly discusses the governing equations in the RANS framework that solve the transport equations for Favre averaged quantities. This is followed by the details of the numerical setup.

3.1. Governing equations

Both VOF and Eulerian multi-fluid (MF) models are commonly used for the simulation of two-phase flows (Soleymani et al. 2008, Parsi et al. 2016, Clissold et al. 2020, Jalalifar et al. 2020, Gallagher et al. 2021). In the Eulerian MF model, momentum, continuity and energy equations are separately solved for each phase (Khezzar et al. 2015), allowing to model various forces, such as lift, drag, and virtual mass. On the other hand, the VOF model solves a single set of momentum equations although it uses a surface-tracking technique designed for two or more immiscible fluids (Chauhan et al. 2021). In VOF, the phase volume fractions are obtained using accurate explicit or implicit time algorithms with higher-order advection schemes to resolve sharp interfaces between a pair of phases. The validation case is simulated using both

approaches, allowing the selection of the proper model for the comprehensive parametric study of the HBC design case.

3.1.1 Eulerian multi-fluid model

The Favre averaged continuity equation for each phase can be written as

$$\frac{1}{\rho_q} \left(\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) \right) = 0 \quad (2)$$

where ρ is density, α is volume fraction, and $\vec{v}$ is velocity. Note q is the primary fluid phase and p is the secondary fluid phase. The Favre averaged momentum equation for each phase is given as

$$\begin{aligned} \frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q \vec{v}_q) \\ = -\alpha_q \nabla p + \nabla \cdot \bar{\tau}_q + \alpha_q \rho_q \vec{g} + \sum_{l=1}^n K_{qp} (\vec{v}_p - \vec{v}_q) + \vec{F}_q + \vec{F}_{lift,q} \\ + \vec{F}_{vm,q} \end{aligned} \quad (3)$$

where p is pressure, $\bar{\tau}$ is the stress-strain tensor, K_{qp} is the fluid-fluid exchange coefficient, $\vec{F}_q$ is external body force, $\vec{F}_{lift}$ is lift force, $\vec{F}_{vm}$ is virtual mass force, and n is the number of phases which is two here.

In the Eulerian multiphase approach, Eqs. (2) and (3) are solved for each phase. Since the dominant interphase force is the drag, the fluid-fluid exchange coefficient, K_{qp} is related to the drag force and relative velocity between phases, and written as

$$K_{qp} = \alpha_q \alpha_p \rho_p f_{drag} / \tau_p \quad (4)$$

In the above equation, $\tau_p = \rho_p d_p^2 / 18 \mu_q$ represents the droplet/bubble response time, where d_p is the droplet/bubble size, and f_{drag} represents the drag function. Various models are available for the drag function. Schiller and Naumann model is acceptable for most fluid-fluid simulations due to its correlation with experimental data across a range of flow velocities and particle/bubble concentrations (Silva et al. 2015). For this model, the drag force and drag coefficient are presented as

$$f_{drag} = \frac{C_D Re_p}{24} \quad (5)$$

$$C_D = \begin{cases} \frac{24}{Re_p} (1 + 0.15 Re_p^{0.687}) & Re_p \leq 1000 \\ 0.44 & Re_p > 1000 \end{cases} \quad (6)$$

where C_D is the drag coefficient and Re_p is the relative Reynolds number, defined in Eq. (7).

$$Re_p = \frac{\rho_q |\vec{v}_p - \vec{v}_q| d_p}{\mu_q} \quad (7)$$

where μ is the viscosity.

3.1.2 Volume of fluid (VOF) coupled level-set

For the VOF method, instead of solving governing equations for each phase, the mass and momentum equations are simplified to one set of equations (Pineda-Pérez et al. 2018). To improve the interface tracking, the VOF is coupled with the level-set method, which is specifically designed for the cases where no mass transfer between phases is involved. The level-set function is smooth and continuous and hence its spatial gradients can be accurately calculated. In this method, the conservation of the mass and momentum are written as

$$\nabla \cdot (\vec{v}) = 0 \quad (8)$$

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot [\mu (\nabla \vec{v} + \nabla \vec{v}^T)] + \rho \vec{g} - \sigma k \delta(\phi) \nabla(\phi) \quad (9)$$

where $\vec{v}^T$ is the stress tensor, ϕ is the level-set function, σ is the surface tension coefficient, and k is the curvature.

The phase interaction between the fluids is defined using the last term at the right-hand side of Eq. (9), which is a function of the level-set function, ϕ . It is defined as a signed distance to the interface and hence the interface is the zero level-set and can be expressed as $\Gamma = \{x | \phi(x, t) = 0\}$ in a two-phase flow system:

$$\phi = \begin{cases} +|d_I| & \text{if } x \in \text{the primary phase} \\ 0 & \text{if } x \in \Gamma \\ -|d_I| & \text{if } x \in \text{the secondary phase} \end{cases} \quad (10)$$

where d_I is the distance from the interface. The normal, $\vec{n}_I$ and curvature, k of the interface is required in the computation of the surface tension force and estimated as

$$\vec{n}_I = \frac{\nabla \phi}{|\nabla \phi|} |_{\phi=0} \quad (11)$$

$$k = \nabla \cdot \frac{\nabla \phi}{|\nabla \phi|} |_{\phi=0} \quad (12)$$

The transport equation of the level-set function is then written as

$$\frac{\partial \phi}{\partial t} + \vec{v} \cdot \nabla \phi = 0 \quad (13)$$

For the secondary phase, the conservation of the volume fraction is also solved which is expressed as:

$$\frac{\partial \alpha_p}{\partial t} + \vec{v} \cdot \nabla \alpha_p = 0 \quad (14)$$

The volume fraction equation for the primary phase is then computed using:

$$\alpha_q = 1 - \alpha_p \quad (15)$$

The presence of the component phases determines the properties appearing in the transport equations. The density and viscosity for the mixture are then computed as

$$\begin{aligned} \rho &= \alpha_q \rho_q + \alpha_p \rho_p \\ \mu &= \alpha_q \mu_q + \alpha_p \mu_p \end{aligned} \quad (16)$$

3.2. Numerical set-up

For the velocity and pressure coupling, the PISO pressure correction algorithm is adopted. The second-order upwind scheme is adopted for the flow parameters. The geo-reconstruct scheme is used for the interface. Residuals are set as 1E-4. The inlet constant velocity is adopted for the water jet. A non-slip condition is imposed at wall boundaries. The top of the tank is open and it is set as a pressure outlet. The standard k- ϵ model is adopted for modelling flow turbulence, which is a semi-empirical linear eddy viscosity model based on the model transport equations for the turbulent kinetic energy and its dissipation rate. Further discussions on selecting turbulence model are given in Section 4.1.

Four different mesh configurations are considered for the case with $d=0.025$ m, $H=2$ m, $D=0.3$ m and $V_0 = 2$ m/s. The near wall meshes are designed to be biased towards the wall so that the wall adjacent cell height can be accurately controlled. The number of cells for these meshes is approximately 6,000, 10,000, 21,400 and 33,800. Figure 2 shows the axial profile of the

velocity at $r=0.075$ m measured from the centre. The velocity is presented for the thin water layer formed at the bottom surface of the column after the jet water impinges on the wall. As can be seen, a configuration with 21,400 cells shows acceptable accuracy in predicting the velocity, and a further refinement results in marginal difference. Hence, a mesh with 21,400 cells is selected for the simulations.

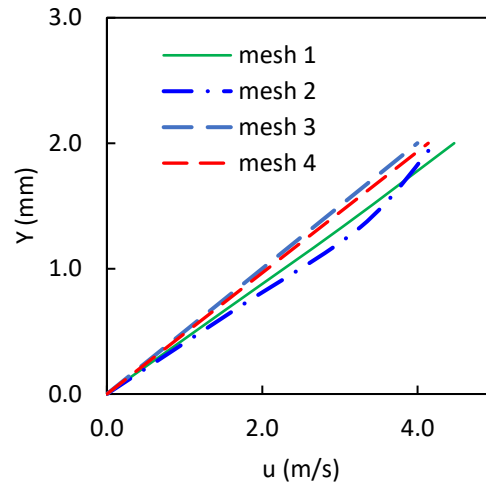


Figure 2: Velocity profiles along the water layer at the bottom of the HBC.

4. Results and discussion

4.1. Model validation

The numerical models are first validated against the experimental data conducted by (Qu et al. 2013). The experiment was performed in a $0.3 \text{ m} \times 0.3 \text{ m} \times 0.5 \text{ m}$ rectangular tank fabricated from transparent acrylic sheets to allow for flow visualisation. The water was injected into the tank by a vertical thin-walled round nozzle with an inner diameter of $d = 6 \text{ mm}$. The free jet falling height is $L_j = 100 \text{ mm}$ which is the distance from the nozzle exit to the water free surface. The water level inside the tank was kept constant at a height of 420 mm measured from the bottom of the tank. The water injection velocity, V_0 , is 1.5 m/s .

Figure 3 shows the time series of the spatial process of initial impact by a plunging jet. It qualitatively compares the results obtained using the VOF level set method and Eulerian MF model with the PIV velocity measurements of the liquid phase. It can be seen that the penetrating jet creates a conical air void under the water surface that grows over time due to the kinetic energy and potential energy balance. The penetration of the water jet continues while the shape is deforming, creating a cylindrical shape at the bottom of the air cavity with the liquid jet trapped inside (Qu et al. 2013). At approximately 50 s , a balance between the local hydrostatic pressure and the initial velocity imparted to the cavity wall causes the rim of

the cylindrical part to collapses at the neck. The bottom part then evolves into a toroidal air bubble. Finally, the toroidal air bubble collapses into several bubbles of various sizes and shapes. This trend is well captured by both VOF level-set and Eulerian MF models although both models show some discrepancies in predicting bubble formation, which is consistent with the previous simulations (Ma et al. 2010).

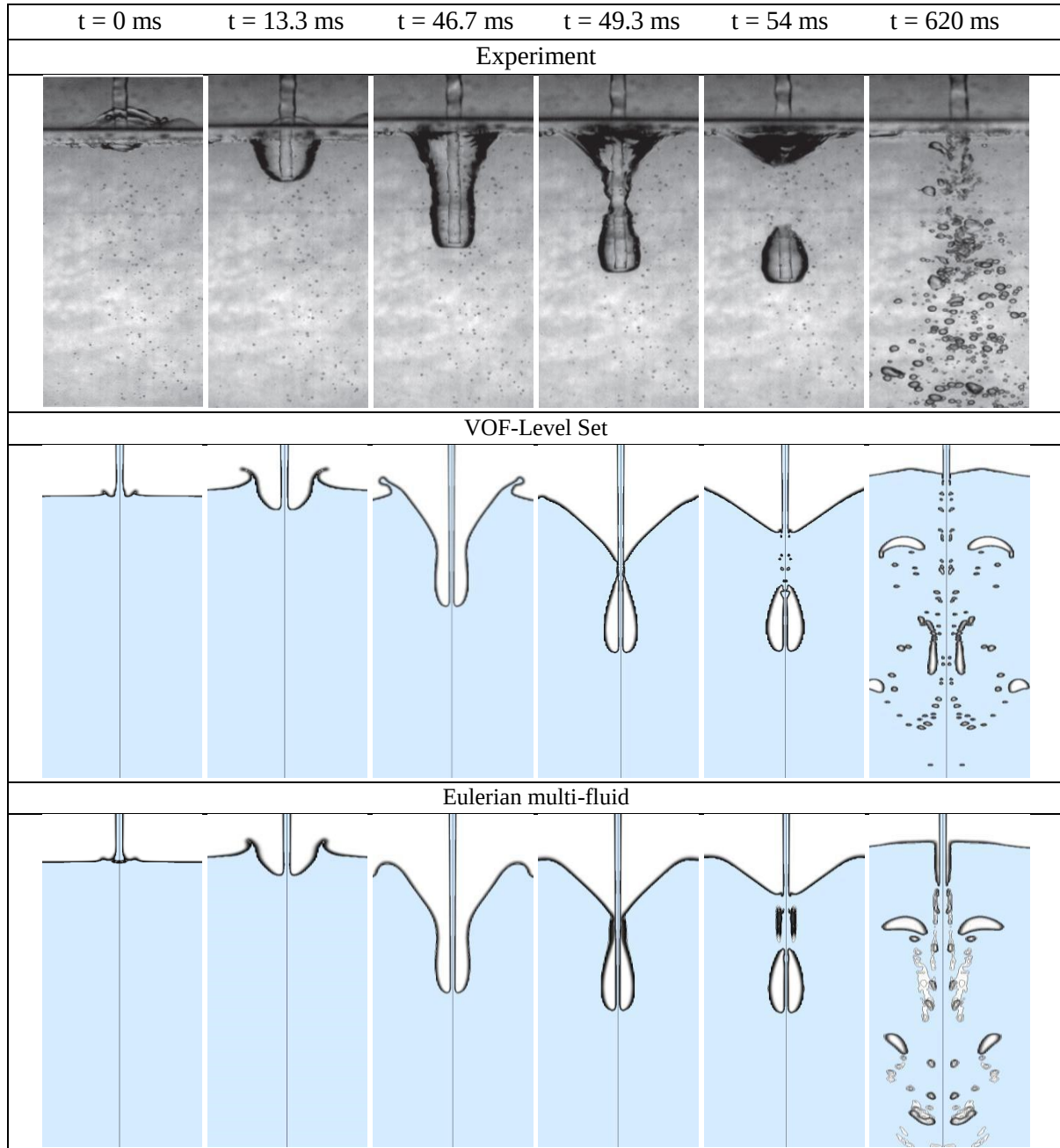


Figure 3: Time series of liquid jet penetration in comparison to the measurements (Qu et al. 2013).

The progress of the initial air cavity can be also monitored using geometrical cavity parameters when the pinch-off occurs, representing when the cavity breaks down into two parts. Three main geometrical parameters are considered here including D_c/d , D_h/d as h_c as schematically

shown in Fig. 4. D_c is the diameter of the approximated hemisphere, D_h is the diameter of the hollow produced by the jet and h_c is the depth of the cavity at pinch-off. These parameters can be also obtained using semi-analytical equations (Zhu et al. 2000):

$$\begin{aligned}\frac{D_c}{d} &\cong 2C(Fr)^{\frac{1}{2}} \\ \frac{D_h}{d} &\cong 2.41 \\ \frac{h_c}{d} &\cong \frac{1}{4}\left(\frac{D_c}{d}\right)^{\frac{2}{3}}(Fr)^{\frac{2}{3}} + 2C(Fr)^{\frac{1}{2}}\end{aligned}\tag{17}$$

where $C = 0.9$ is a constant and Fr is the Froude number and computed as $Fr = V_0/\sqrt{gd}$. Note that Fr is 6.183 for the validation case.

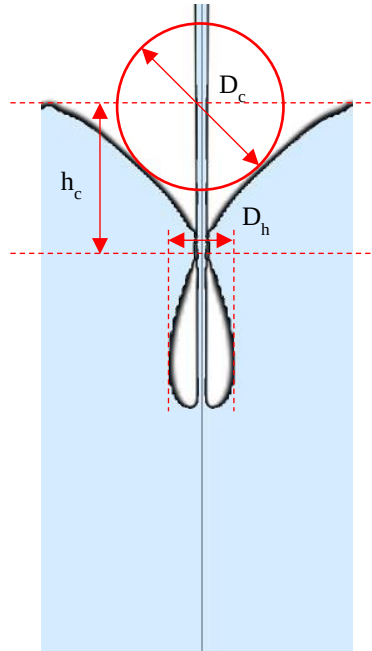
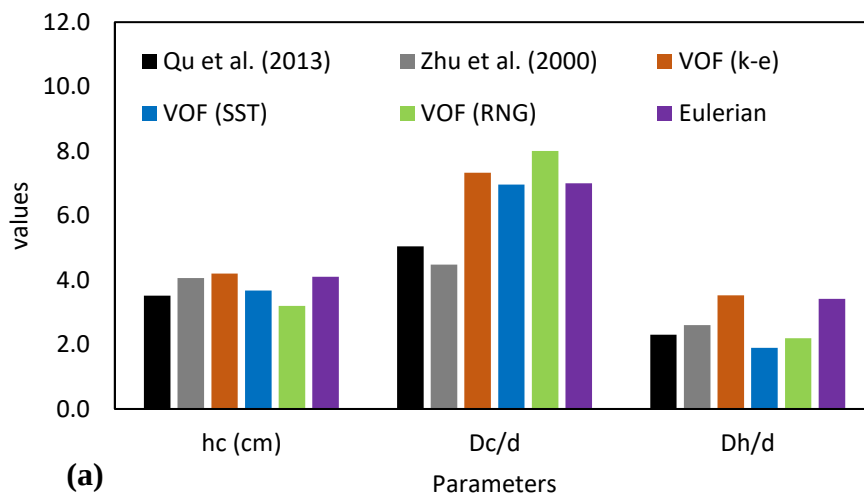


Figure 4: Schematic view of the geometric parameters of the cavity.

The simulation results are compared with the measured and semi-analytical data (Zhu et al. 2000, Qu et al. 2013) in Fig. 5a. It can be seen that both VOF and Eulerian MF models predicted the depth of the cavity very well. The agreement for D_h/d is reasonable while D_c/d is over predicted which is consistent with the observation in Fig. 3 at approximately 50 s where the pinch-off occurs. The velocity of the cavity bottom, V_c , that represents the penetration velocity is further analysed in Fig. 5b. The penetration velocity is normalised by the relevant jet velocity, V_j , which is computed using Eq. (1). The cavity depth y_c is normalised by the nozzle diameter.

Both models agree well with the experimental data. It can be seen that, when the cavity is produced by a steady liquid jet, the bottom of the cavity advances at a velocity approximately equal to half the jet velocity (Birkhoff 1912).

In addition to the standard $k-\epsilon$ model, the VOF simulations were conducted using more advanced turbulence models including the $k-\epsilon$ renormalisation group turbulence model (RNG) and $k-\omega$ based shear stress transport (SST) model which are expected to have better accuracy near-wall region (Castillo and Carrillo 2017, Castillo et al. 2017). As can be seen in Fig. 5, the results reveal that the SST and RNG models do not necessarily provide better agreement with the measured data. While the SST model shows slightly better agreement for h_c , it leads to underprediction of D_h/d . On the other hand, the RNG model resulted in the underprediction of h_c and D_h/d whereas it slightly improved the prediction of D_c/d . The results obtained using these turbulence models for the normalised penetration velocity show the same level of agreement as that obtained using the standard $k-\epsilon$ model. Hence, the $k-\epsilon$ model is used for the simulations in this study. It is worth noting that for the hydraulic break column case, the near wall region does not significantly affect the air entrainment during the filling process which is the focus of the current study and hence the $k-\epsilon$ model should be adequate for the simulations.



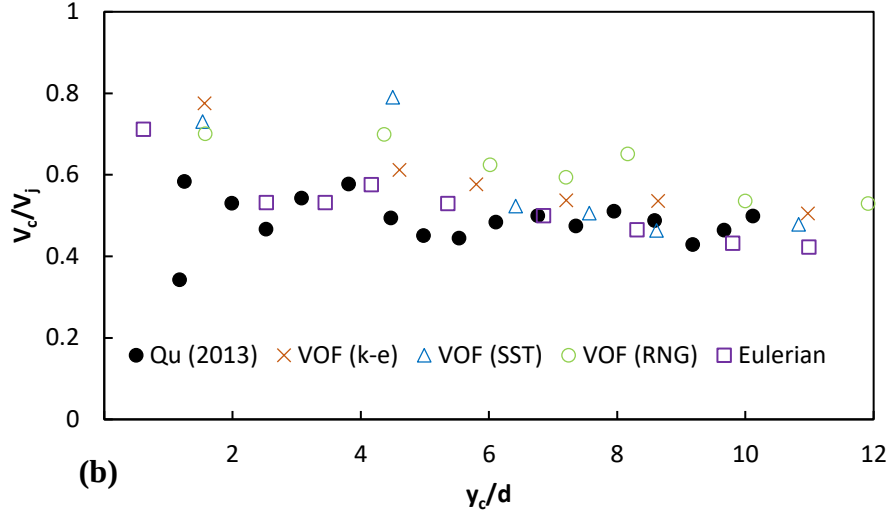


Figure 5: Model validation: (a) geometrical cavity parameters, (b) normalised penetration velocity V_c/V_j versus normalised cavity depth y_c/d .

The results for the validation case confirm that both VOF and Eulerian MF models are capable to accurately predict air entrainment in a two-phase plunging jet flow. Hence, for the HBC simulations, the VOF model is adopted which is computationally more affordable.

4.2. Hydraulic break columns

This section provides VOF simulations of the filling process of the HBC of Fig. 1. The geometrical and operational parameters vary according to the values of Table 1. In total 26 simulation cases are considered. The aim is to understand how these parameters affect the air entrainment during the filling process in order to identify design guidelines for an HBC.

Figures 6 and 7 show the temporal evolution of water in the HBC for a series of selected cases. For an ideal scenario where there is no air entrainment, it is expected that the water level, y in the tank increases linearly with time. Depending on the air entrainment, the water level deviates from the theoretical values as shown in Fig. 6. In this figure, the theoretical values are presented by dashed lines, while the actual values, obtained using the CFD simulations, are presented by solid lines. Three main patterns are identified among different cases: dispersed bubbles, medium air pockets (MAP) and large air pockets (LAP). For more clarification, a time series of the water volume fraction is also shown in Fig. 7 for both bubble and MAP conditions. It can be seen that water reaches the impinging wall at the bottom surface of the column and then it spreads horizontally until it reaches the sidewalls. As expected, the water then moves

upwards until it reaches the height of h where its upward velocity becomes zero. The value of h is greater for the case with a higher V_0 as shown in Fig. 7a and Fig 7d. This is the reason that, in all three recognised patterns, the water level initially deviates sharply from the theoretical values. The water then fountains back to the tank due to the gravitational effect (Fig. 7b and Fig. 7e) and hence the trapped air creates some air pockets. It also should be noted that during the return of the fountain, in some cases, it clashes with the incoming jet disturbing the impinging process that causes a more violent beginning in the filling process.

In the first scenario, referred to as the *bubble case*, the initially formed air pockets break into smaller bubbles (Fig. 7b and Fig. 7c) and hence after a violent start, the rise of water level travels in parallel to the theoretical values. This is indicated by blue solid and dashed lines in Fig. 6. In this case, the air bubbles are formed under the water surface at a constant linear rate but slightly above the theoretical values indicating the portion of air trapped in the volume. In the second scenario, referred to as the *MAP case*, the size of air pockets is larger compared to the bubble case as shown in Fig. 7e and Fig. 7f. However, after some time they burst at the surface and water column drops, allowing the filling process to almost return to the bubble air entrainment regime. Hence, eventually, the water level increases linearly as shown by green solid and dashed lines in Fig. 6. Finally, in the third scenario, referred to as the *LAP case*, the formed air pockets are very large. Due to the higher energy of the impinging jet combined with a smaller tank diameter, the inkling jet interacts violently with the wall and hence the returning flow at the beginning of the filling process creates large air pockets that do not burst due to the fast accumulation of water on top of the large air pockets. Consequently, the water level quickly reaches its maximum value (i.e. the height of the tank), causing an overflow.

By varying either the geometrical parameters or initial velocity, one of the discussed scenarios may occur. Among the mentioned scenarios, the bubble condition is considered as operational, while the LAP case is considered as the failed scenario. Although for the MAP case, the water does not overflow, this condition is not reliable since the flow behaviour is violent and should preferably be prevented. All cases are categorised into the bubble, MAP and LAP based on the observed trends for the water level rise and discussed in more detail in the following.

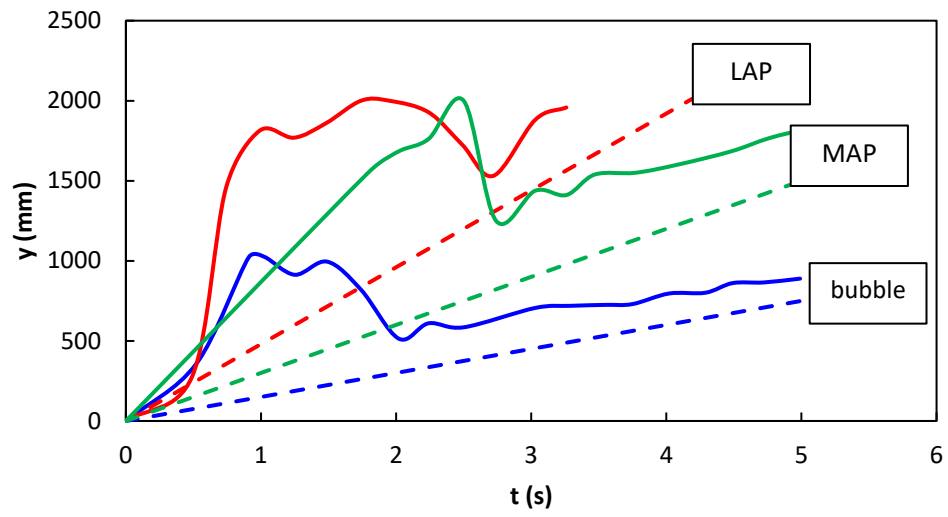


Figure 6: Temporal evolution of water level for three main scenarios: dispersed bubble, medium air pocket (MAP) and large air pocket (LAP).

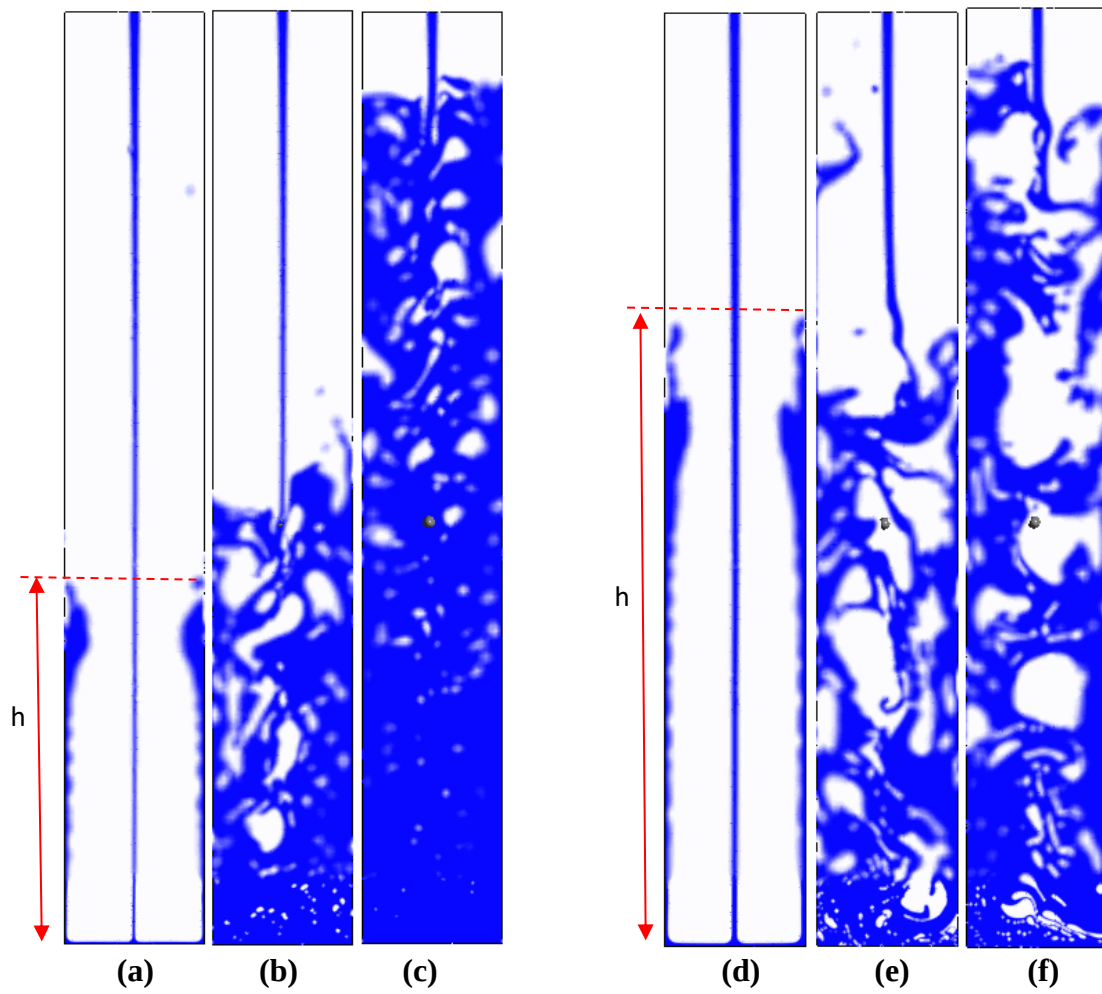
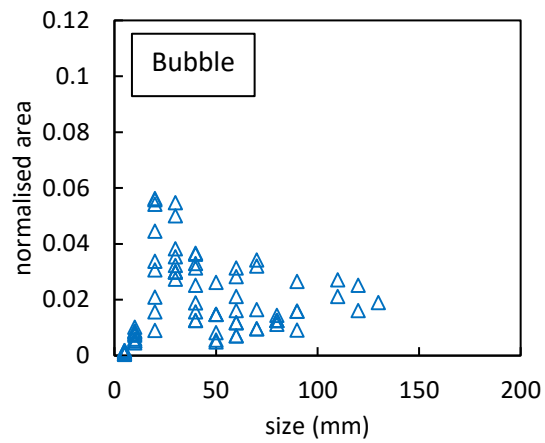


Figure 7: Time series of water volume fraction for the case with $V_0 = 2$ m/s, representing the bubble pattern at (a) 1 s (b) 4.86 s, and (c) 9.94 s and the case with $V_0 = 4$ m/s, representing the MAP pattern at (d) 0.93 s, (e) 2.51 s, and (f) 3.56 s.

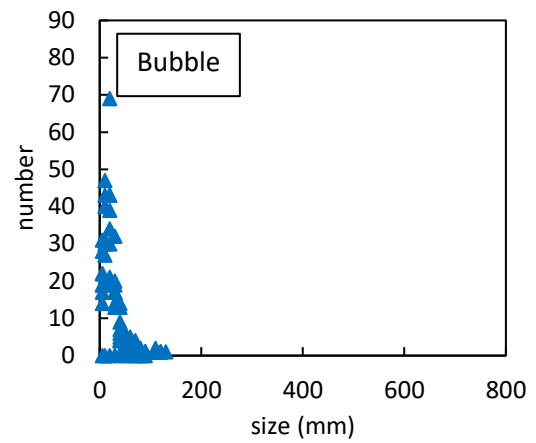
To provide some information on the bubble and air pocket size, the image processing of 2D plots of volume fractions is conducted using IMAGEJ. For all cases, the bubbles are recognised at the time when the water level reaches 80% of the column's height. The size distribution is then obtained to compute the normalised total area that represents the total volume fraction of air.

$$\alpha_i = N_i A_i / A_{total} \quad (18)$$

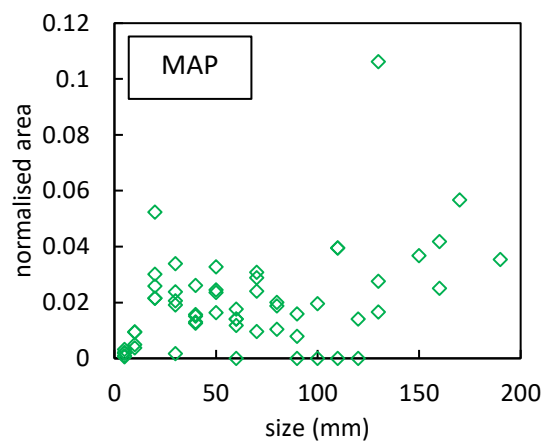
where A_i and N_i are the area and the number of bubble/air pocket i while A_{total} is computed as $0.8D \times H$. Figure 8 presents the probability distribution function of the normalised total area (Fig. 8a, c, e) and the number of bubble/air pockets (Fig. 8b, d, f) versus the bubble equivalent diameter. For all different patterns, the frequency of small bubbles is the highest, however, the peak of normalised area distribution moves towards larger sizes as the flow pattern varies from bubble to LAP. While in MAP and LAP scenarios, the number of larger bubble/air pocket sizes is smaller than that of the small bubbles, their areas (representing their volume fraction) are significantly larger that changes the trend of water rise in the column. For the cases with bubble patterns, the bubble size ranges from approximately 5 mm to 130 mm while their total area ranges from 0.1% to 5.6% of the column. Small bubbles with low volume fractions allow the water level to increase almost linearly. For MAP cases, the air pocket's size range is further extended, showing a maximum of the equivalent diameter of 190 mm while the maximum normalised bubble/air pockets area is about 10% of the tank volume. It confirms that the water level still follows the linear trend although the flow is more violent due to the existence of the larger bubbles. For the LAP scenarios, the large air pockets are created - a maximum size of 590 mm which occupied about 60% of the column, causing water overflow to occur in these cases. It is worth noting that there are some scatter for larger bubble sizes which could be due to the limited number of available samples.



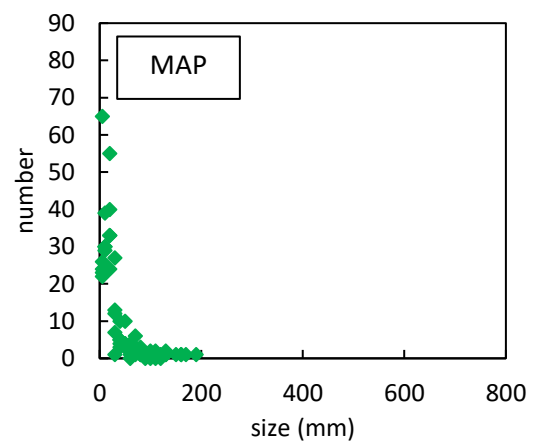
(a)



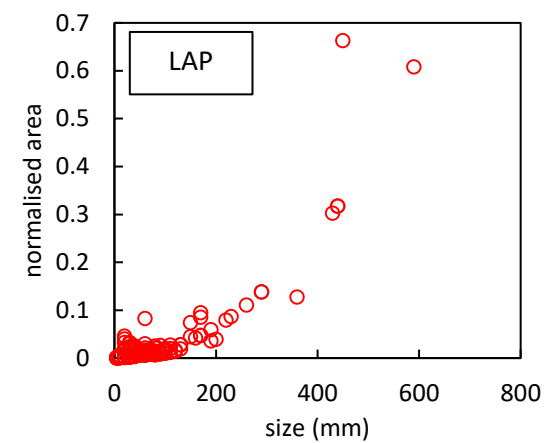
(b)



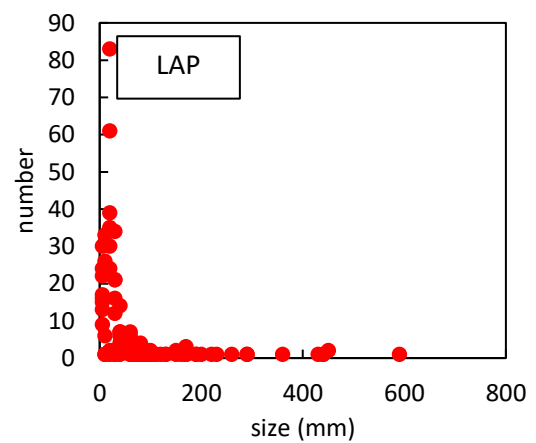
(c)



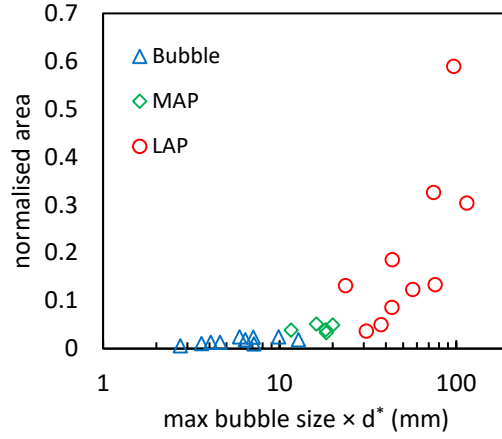
(d)



(e)



(f)



(g)

Figure 8: (a,c,e) Normalised bubble area and (b,d,f) bubble number versus bubble diameter, and (g) normalised bubble area versus maximum bubble equivalent diameter.

Furthermore, Fig. 8g shows the maximum bubble/air pocket equivalent diameter which is scaled by d^* defined as the ratio of nozzle diameter to the tank diameter

$$d^* = d/D \quad (19)$$

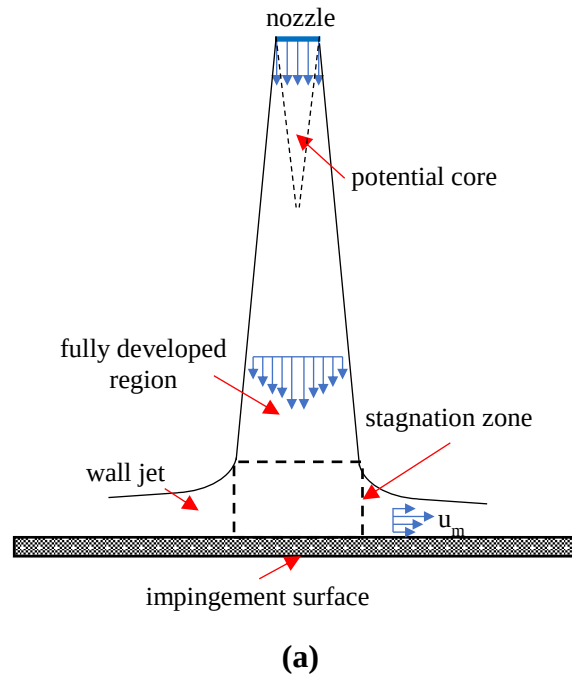
It can be seen that the scaled equivalent air bubble diameter is the lowest for bubble cases. It shows a medium range for MAP cases while its value is significantly higher for LAP cases. For the LAP cases, a larger area of the tank is occupied by the air pockets trapped in the initial stage of the filling and hence the water level sharply increases in the early stage. For two cases that have the same nozzle diameter and jet velocity but different tank diameters, the narrower tank shows that the initial entrappings of large pockets did not have enough time to raise by its buoyancy before overflowing occurs. As the tank diameter widens, the returning fountain interacts violently with the water jet allowing the entrained pockets to break and hence smaller air pockets and bubbles are formed. Consequently, the overflowing process is postponed and the water level increasing rate returns to a linear trend as observed in Fig. 6 for bubble cases. The size of bubbles and air pockets are correlated to the operating conditions and geometrical features of the column that can be represented by non-dimensional numbers such as Frude and Reynolds numbers which are discussed further in the following.

The non-dimensional Reynolds and Froude numbers are obtained using

$$Re_j = \rho_w V_j / \mu_w d$$

$$Fr = \frac{V_j}{\sqrt{gd}} \quad (20)$$

where ρ_w and μ_w are the density and viscosity of water, respectively. V_j is obtained using Eq (1) where $L_j = H$. The jet Reynolds number describes the relative importance of inertial forces and viscous forces in the jet whereas the Froude number represents the inertial forces and gravitational forces on the surface of the jet. The formation of bubbles is directly linked to the flow pattern of the jet impinging. To further shed light on this, a schematic view of different regions of a jet impingement flow is shown in Fig. 9a while the velocity field is analysed in Fig. 9b for two cases with $V_0 = 2$ m/s and 4 m/s. In both cases, $d=0.025$ m, $D=0.3$ m and $H=2$ m. At the nozzle exit, entrainment of mass, momentum and energy occurs due to shear-driven interaction of the surroundings with the existing jet. Gradually, the jet loses its kinetic energy and hence the velocity profile expands in the spatial direction while shrinks down in magnitude along the sides of the jet. This leads to a non-uniform radial profile of the axial velocity that develops within the jet. This is consistent with the previous studies on the impinging jet (Dewan et al. 2012, Yang et al. 2015). The point where the jet strikes the surface is known as the deflection zone/stagnation region. In this region, the axial velocity quickly diminishes with a consequent rise in the static pressure (Ahmed et al. 2015). The bulk flow then follows the radial outward direction, creating the wall jet which is a layer of water at the bottom surface of the column. This is followed by the fountain region along the sidewalls, which is caused by the interaction of water flow on the bottom surface with the sidewalls.



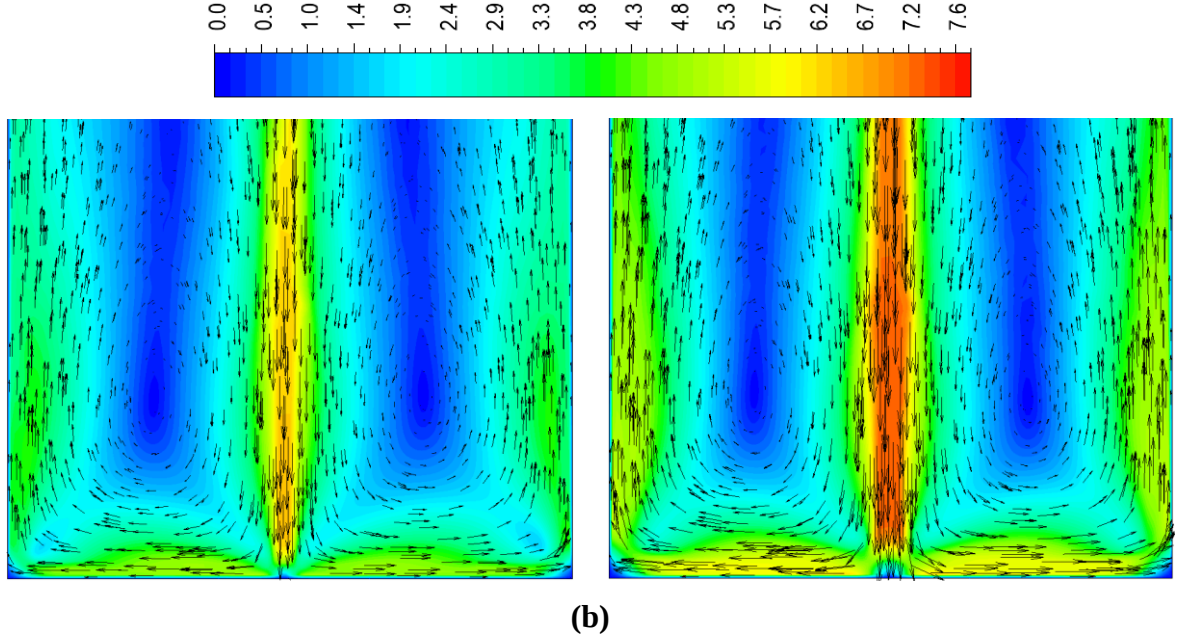


Figure 9: Jet impingement flow: (a) Schematic view of different regions, and (b) velocity contour plots and vectors near the bottom of the column for (left) the case with $V_0 = 2$ m/s, representing the bubble pattern and (right) the case with $V_0 = 4$ m/s, representing the MAP pattern.

In the wall jet zone, the axial profile of the radial velocity is important. At a radial location, the radial velocity is zero adjacent to the impinging wall and increases and reaches its maximum value which is shown by u_m in Fig. 9a. Further away from the centre of the tank, the value of u_m is smaller. Hence, as radially moving toward the tank sidewalls, u_m decreases, showing that the maximum u_m occurs near the centreline of the column, shown by $u_{m,max}$. As expected the case with higher V_0 has a higher jet velocity, resulting in a larger $u_{m,max}$ at the wall jet zone (Yang et al. 2015). Figure 10 illustrates the $u_{m,max}$ normalised by V_j versus the jet Reynolds number scaled with d^* for all cases. It can be seen that the normalised $u_{m,max}$ fluctuates around a value of 0.87, marked by a horizontal dashed line, showing a minimum and maximum value of 0.72 and 0.95, respectively. The cases with lower $Re_j \times d^*$ dominantly result in lower $\frac{u_{m,max}}{V_j}$, while the opposite is true for the cases with larger $Re_j \times d^*$.

This is due to the development of the jet core before it reaches the wall surface. A case with a smaller nozzle diameter has a lower value of Re_j and d^* (i.e d/D) and consequently lower $Re_j \times d^*$. In this case, the jet core zone is narrower and hence to maintain the flow the energy lost is higher as the jet impinges on the surface, resulting in lower $u_{m,max}/V_j$, whereas for a case with larger nozzle diameter (i.e. larger $Re_j d^*$), the core rejoin is wider and hence the

energy loss is lower when the velocity changes its direction from vertical to horizontal, leading to $u_{m,max}/V_j$ closer to 1. This is shown as an inset in Fig. 10, representing $u_{m,max}/V_j$ for the cases with $D=0.3$ m, $H=2$ m and $V_0 = 2$ m/s whereas the nozzle diameter varies (0.025, 0.05 and 0.08 m).

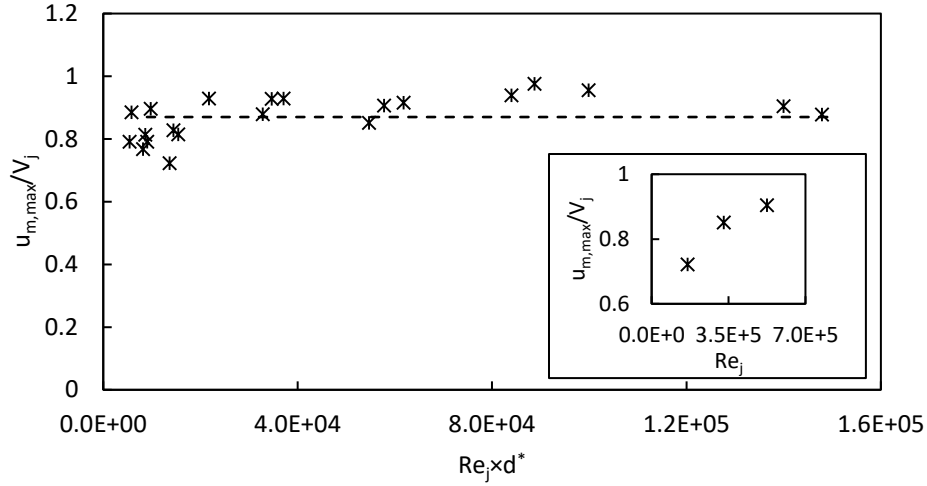


Figure 10: The maximum wall jet velocity versus jet Reynolds number.

Figure 11 presents h/D (open symbols) versus $Re_j \times d^*$. As indicated in Fig. 7, h is the fountain height. The value of D/d for all cases is also presented by filled symbols in this figure. It can be seen that for small jet Reynolds numbers, the flow regime remains in the bubble category. The jet Reynolds number, computed per Eq. (20), is affected by V_j which depends on the column's height (H) and the water velocity at the nozzle exit (V_0). Hence, low Re_j values represent the cases with lower initial flux and smaller column's height (H) that affects the formation of bubbles. These cases also have a high D/d ratio where the jet momentum and tank size allow the trapped air to break down into small bubbles and hence the water level increases linearly. On the other hand, the cases with large Re_j and small D/d dominantly follow the LAP category where large air pockets remain in the domain, causing the filling process to be failed. The MAP cases are characterised by medium Re_j while their D/d remains in the mid-range.

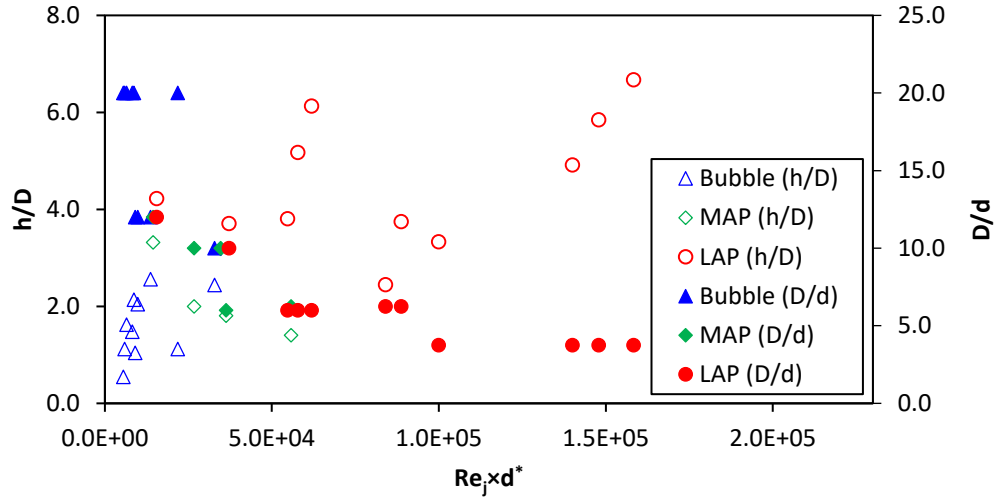


Figure 11: The normalised fountain height (h/D) and the ratio of the tank diameter to the nozzle diameter (D/d) versus the jet Reynolds number.

The relationship between the Froude and jet Reynolds numbers is further analysed in Fig. 12 for different cases. Both non-dimensional numbers are scaled by d^* . It can be seen that for the cases with large Re_j and Fr numbers, the LPA phenomena is dominant whereas, for cases with small Re_j and Fr numbers, the bubble regime is dominant which is considered as operational. This provides a guideline for selecting the appropriate operating conditions and HBC geometry features. The overflow condition can be avoided if the combination of d , D , H , and V_0 results in small values of $Fr \times d^*$ and $Re_j \times d^*$ as shown in Fig. 12.

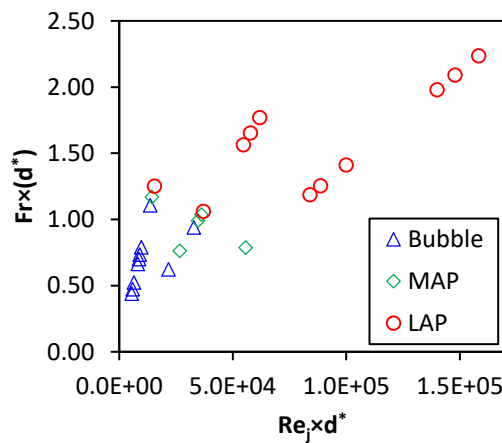


Figure 12: Dependence of air entrainment regimes on Froude and Reynolds numbers

5. Conclusions

This paper presented detailed computational fluid dynamic (CFD) simulations of air entrainment phenomena occurring during the filling process of a hydraulic break column (HBC). The CFD model was first validated against the measurement data for water plunging jet into a water pool. For this purpose, both volume-of-fluid (VOF) and Eulerian multi-fluid models were tested. It was found that both models accurately predicted the geometric parameters of the cavity at the pinch-off point. Satisfactory agreements were also obtained between the CFD data and measurements for both penetration length and penetration velocity.

The VOF model was then adopted for the HBC simulation. Effects of inlet jet velocity, nozzle diameter, tank diameter, and tank height were numerically analysed. It was found that the air entrainment rate is determined by a combination of impinging jet and water plunging physics and is affected by the operating and geometrical parameters. Among different cases, three main patterns are identified which are namely dispersed bubble, medium air pockets (MAP) and large air pockets (LAP) scenarios. In the bubble case, the formed air pockets break into smaller size bubbles and hence the water level rises linearly. In the MAP scenario, the size of air pockets is larger compared to the bubble case, however, after some time they burst at the surface and water column drops, allowing the filling process to almost return to the bubble air entrainment regime. Finally, in the LAP scenario, the impinging jet interacts violently with the wall and hence large air pockets are developed. For HBCs forming dispersed bubbles, the process is operational, while for cases demonstrating other patterns (MAP and LAP), the flow behaviour is trapping more air in the column and hence they should preferably be prevented.

The complex two-phase flows are further analysed using non-dimensionless numbers, including the ratio of the jet and tank diameters, the normalised fountain height formed on the sidewall surfaces and Reynolds and Froude numbers. It was found that at lower Reynolds numbers and higher ratios of jet to tank diameter, the normalised fountain height is lower and hence there is enough time and space for the formed air pockets to burst, leading to the bubble regime. Further analysis showed that there was a correlation between Reynolds and Froude numbers, showing that for low values of these parameters, the flow regime remains in the dispersed bubble category where the water level in the column rises linearly with time similar to the ideal theoretical condition.

<i>Nomenclature</i>		
A_i	(m ²)	Area of bubble/air pocket
C_D	-	Drag coefficient
d	(m)	Nozzle diameter
D	(m)	Tank diameter
D_c	(m)	Diameter of the hemisphere
D_h	(m)	Diameter of the hollow
d_p	(m)	Bubble size
d^*	-	Non-dimensional diameter
d_I	(m)	Distance from the interface
F_q	N	External body force
F_{lift}	N	Lift force
F_{vm}	N	Virtual mass force
f_{drag}	N	Drag force
Fr	-	Froude number
g	(m/s ²)	Gravity
h	(m)	Fountain height
H	(m)	Tank height
h_c	(m)	Depth of the cavity at pinch-off
k	(1/m)	Curvature
K_{qp}	(kg/m ³ s)	Fluid-fluid exchange coefficient
L_j	(m)	Falling length
N_i	-	number of bubble/air pocket
p	(Pa)	Pressure
r	(m)	Tank radius
Re_j	-	Jet Reynolds number
Re_p	-	Bubble Reynolds number
u_m	(m/s)	Water velocity at the bottom of the tank
V_0	(m/s)	Water velocity at the nozzle exit
V_j	(m/s)	Impact jet velocity
V_c	(m/s)	The velocity of the cavity bottom
y	(m)	Water level
y_c	(m)	Cavity depth
α	-	Volume fraction
μ	(kg/m.s)	Viscosity
ρ	(kg/m ³)	Density
σ	(kg/m ² s ²)	Surface tension coefficient
$\bar{\tau}$	(kg/ms ²)	Stress-strain tensor
τ_p	(t)	Bubble response time
ϕ	(m)	Level-set function

References

- Ahmed, Z. U., Y. M. Al-Abdeli and F. G. Guzzomi (2015). "Impingement pressure characteristics of swirling and non-swirling turbulent jets." Experimental Thermal and Fluid Science **68**: 722-732.
- Bhusare, V. H., M. K. Dhiman, D. V. Kalaga, S. Roy and J. B. Joshi (2017). "CFD simulations of a bubble column with and without internals by using OpenFOAM." Chemical Engineering Journal **317**: 157-174.
- Biń, A. K. (1993). "Gas entrainment by plunging liquid jets." Chemical Engineering Science **48**(21): 3585-3630.
- Birkhoff, G. (2012). Jets, wakes, and cavities, Elsevier.
- Brouilliot, D. and P. Lubin (2013). "Numerical simulations of air entrainment in a plunging jet of liquid." Journal of Fluids and Structures **43**: 428-440.
- Castillo, L. G. and J. M. Carrillo (2017). "Comparison of methods to estimate the scour downstream of a ski jump." International Journal of Multiphase Flow **92**: 171-180.
- Castillo, L. G., J. M. Carrillo and F. A. Bombardelli (2017). "Distribution of mean flow and turbulence statistics in plunge pools." Journal of Hydroinformatics **19**(2): 173-190.
- Cervantes-Álvarez, A. M., Y. Y. Escobar-Ortega, A. Sauret and F. Pacheco-Vázquez (2020). "Air entrainment and granular bubbles generated by a jet of grains entering water." Journal of Colloid and Interface Science **574**: 285-292.
- Chauhan, A., F. Salehi, S. Jalalifar and S. M. Clark (2021). "Two-phase modelling of the effects of pore-throat geometry on enhanced oil recovery." Applied Nanoscience.
- Chirichella, D., R. Gomez Ledesma, K. Kiger and J. Duncan (2002). "Incipient air entrainment in a translating axisymmetric plunging laminar jet." Physics of Fluids **14**(2): 781-790.
- Clissold, J., S. Jalalifar, F. Salehi, R. Abbassi and M. Ghodrat (2020). "Fluidisation characteristics and inter-phase heat transfer on product yields in bubbling fluidised bed reactor." Fuel **273**: 117791.
- Dewan, A., R. Dutta and B. Srinivasan (2012). "Recent trends in computation of turbulent jet impingement heat transfer." Heat Transfer Engineering **33**(4-5): 447-460.
- El Hammoumi, M., J.-L. Achard and L. Davoust (2002). "Measurements of air entrainment by vertical plunging liquid jets." Experiments in fluids **32**(6): 624-638.
- Gallagher, C., S. Jalalifar, F. Salehi, A. Kourmatzis and S. Cheng (2021). "A two-fluid model for powder fluidisation in turbulent channel flows." Powder Technology **389**: 163-177.
- Guo, K., T. Wang, Y. Liu and J. Wang (2017). "CFD-PBM simulations of a bubble column with different liquid properties." Chemical Engineering Journal **329**: 116-127.
- Jalalifar, S., M. Masoudi, R. Abbassi, V. Garaniya, M. Ghiji and F. Salehi (2020). "A hybrid SVR-PSO model to predict a CFD-based optimised bubbling fluidised bed pyrolysis reactor." Energy **191**: 116414.
- Karim, I., S. V. Ghatage, M. J. Sathe, J. B. Joshi and G. M. Evans (2017). "Instabilities due to turbulence through inlet jet in plunging jet bubble column." Chemical Engineering Science **157**: 76-87.
- Khezzar, L., N. Kharoua and K. T. Kiger (2015). "Large eddy simulation of rough and smooth liquid plunging jet processes." Progress in Nuclear Energy **85**: 140-155.
- Kiger, K. T. and J. H. Duncan (2012). "Air-entrainment mechanisms in plunging jets and breaking waves." Annual Review of Fluid Mechanics **44**: 563-596.
- Kulkarni, A. L. and A. W. Patwardhan (2014). "CFD modeling of gas entrainment in stirred tank systems." Chemical Engineering Research and Design **92**(7): 1227-1248.
- Lau, Y. M., K. T. Sujatha, M. Gaeini, N. G. Deen and J. A. M. Kuipers (2013). "Experimental study of the bubble size distribution in a pseudo-2D bubble column." Chemical Engineering Science **98**: 203-211.
- Lopes, P., G. Tabor, R. F. Carvalho and J. Leandro (2016). "Explicit calculation of natural aeration using a Volume-of-Fluid model." Applied Mathematical Modelling **40**(17): 7504-7515.

Ma, J., A. Oberai, D. Drew, R. Lahey Jr and F. Moraga (2010). "A quantitative sub-grid air entrainment model for bubbly flows–plunging jets." Computers & Fluids **39**(1): 77-86.

Majidi, S. H. and C. Beckermann (2019). "Effect of pouring conditions and gating system design on air entrainment during mold filling." International Journal of Metalcasting **13**(2): 255-272.

Majumder, S. K. (2008). "Analysis of dispersion coefficient of bubble motion and velocity characteristic factor in down and upflow bubble column reactor." Chemical engineering science **63**(12): 3160-3170.

Majumder, S. K., G. Kundu and D. Mukherjee (2006). "Bubble size distribution and gas–liquid interfacial area in a modified downflow bubble column." Chemical Engineering Journal **122**(1): 1-10.

Mandal, A., G. Kundu and D. Mukherjee (2005). "A Comparative Study of Gas Holdup, Bubble Size Distribution and Interfacial Area in a Downflow Bubble Column." Chemical Engineering Research and Design **83**(4): 423-428.

Metzger, L. and M. Kind (2015). "On the transient flow characteristics in Confined Impinging Jet Mixers - CFD simulation and experimental validation." Chemical Engineering Science **133**: 91-105.

Miwa, S., T. Moribe, K. Tsutsumi and T. Hibiki (2018). "Experimental investigation of air entrainment by vertical plunging liquid jet." Chemical Engineering Science **181**: 251-263.

Okhovat-Alavian, S. M., J. Shabani, H. R. Norouzi, R. Zarghami, J. Chaouki and N. Mostoufi (2019). "Effect of interparticle force on gas dynamics in a bubbling gas–solid fluidized bed: A CFD-DEM study." Chemical Engineering Research and Design **152**: 348-362.

Parsi, M., M. Agrawal, V. Srinivasan, R. E. Vieira, C. F. Torres, B. S. McLaury, S. A. Shirazi, E. Schleicher and U. Hampel (2016). "Assessment of a hybrid CFD model for simulation of complex vertical upward gas–liquid churn flow." Chemical Engineering Research and Design **105**: 71-84.

Pineda-Pérez, H., T. Kim, E. Pereyra and N. Ratkovich (2018). "CFD modeling of air and highly viscous liquid two-phase slug flow in horizontal pipes." Chemical Engineering Research and Design **136**: 638-653.

Qu, X., A. Goharzadeh, L. Khezzar and A. Molki (2013). "Experimental characterization of air-entrainment in a plunging jet." Experimental Thermal and Fluid Science **44**: 51-61.

Ramirez de la Torre, R. G., M. Kuchta and A. Jensen (2020). "Experiments on air entrainment produced by a circular free falling jet." International Journal of Multiphase Flow **132**: 103424.

Roy, A., B. Maiti and P. Das (2013). "Visualisation of air entrainment by a plunging jet." Procedia Engineering **56**: 468-473.

Schulz, J. and H.-J. Bart (2019). "Analysis of entrained liquid by use of optical measurement technology." Chemical Engineering Research and Design **147**: 624-633.

Silva, R., F. A. Garcia, P. M. Faia and M. G. Rasteiro (2015). "Evaluating the performance of the mixture model coupled with high and low Reynolds turbulence closures in the numerical description of concentrated solid-liquid flows of settling particles." The Journal of Computational Multiphase Flows **7**(4): 241-257.

Soleymani, A., A. Laari and I. Turunen (2008). "Simulation of drop formation in a single hole in solvent extraction using the volume-of-fluid method." Chemical Engineering Research and Design **86**(7): 731-738.

Torré, J.-P., D. F. Fletcher, T. Lasuye and C. Xuereb (2008). "An experimental and CFD study of liquid jet injection into a partially baffled mixing vessel: A contribution to process safety by improving the quenching of runaway reactions." Chemical Engineering Science **63**(4): 924-942.

Wang, H., N. S. Slamet, G. Zhang and H. Chanson (2018). "Intrusive measurements of air-water flow properties in highly turbulent supported plunging jets and effects of inflow jet conditions." Chemical Engineering Science **177**: 245-260.

Xiang, M., S. Cheung, J. Tu and W. Zhang (2014). "A multi-fluid modelling approach for the air entrainment and internal bubbly flow region in hydraulic jumps." Ocean engineering **91**: 51-63.

Yang, Y.-T., Y.-H. Wang and J.-C. Hsu (2015). "Numerical thermal analysis and optimization of a water jet impingement cooling with VOF two-phase approach." International Communications in Heat and Mass Transfer **68**: 162-171.

Zhu, Y., H. N. Oğuz and A. Prosperetti (2000). "On the mechanism of air entrainment by liquid jets at a free surface." Journal of Fluid Mechanics **404**: 151-177.

Zidouni Kendil, F., D. V. Danciu, M. Schmidtke, A. Bousbia Salah, D. Lucas, E. Krepper and A. Mataoui (2012). "Flow field assessment under a plunging liquid jet." Progress in Nuclear Energy **56**: 100-110.