Characterization of plunging liquid jets: A combined experimental and numerical investigation

X.L. Qu, L. Khezzar, D. Danciu, M. Labois, D. Lakehal

Abstract
This paper presents a combined experimental and numerical study of the flow characteristics of round vertical liquid jets plunging into a cylindrical liquid bath. The main objective of the experimental work consists in determining the plunging jet flow patterns, entrained air bubble sizes and the influence of the jet velocity and variations of jet falling lengths on the jet penetration depth. The instability of the jet influenced by the jet velocity and falling length is also probed. On the numerical side, two different approaches were used, namely the mixture model approach and interface-tracking approach using the level-set technique with the standard two-equation turbulence model. The numerical results are contrasted with the experimental data. Good agreements were found between experiments and the two modelling approaches on the jet penetration depth and entraining flow characteristics, with interface tracking rendering better predictions. However, visible differences are observed as to the jet instability, free surface deformation and subsequent air bubble entrainment, where interface tracking is seen to be more accurate. The CFD results support the notion that the jet with the higher flow rate thus more susceptible to surface instabilities, entrains more bubbles, reflecting in turn a smaller penetration depth as a result of momentum diffusion due to bubble concentration and generated fluctuations. The liquid average velocity field and air concentration under tank water surface were compared to existing semi-analytical correlations. Noticeable differences were revealed as to the maximum velocity at the jet centreline and associated bubble concentration. The mixture model predicts a higher velocity than the level-set and the theory at the early stage of jet penetration, due to a higher concentration of air that cannot rise to the surface and remain trapped around the jet head. The location of the maximum air content and the peak value of air holdup are also predicted differently.

1. Introduction

In plunging liquid jets the flow consists of an evolving liquid column passing through an ambient atmosphere before impinging on the free surface of a receiving liquid bath. Depending on the flow Reynolds number and height of the falling jet, a bubble swarm may be entrained beneath the surface by the impacting jet. This mechanism is frequently encountered in a number of practical situations, such as the aeration of lakes and in waste water treatment, mixing of stagnant water, de-stratification of water reservoirs, and in nuclear and metallurgical industries. Depending on the application considered, the flow characteristics can play a desired, positive role, or in contrast an undesired, negative role. Plunging jets are found to be particularly necessary in the manufacturing of atomizers and emergency cooling techniques in the nuclear industry.

However, in equipment such as chemical reactors, a plunging liquid jet may give rise to bubbly plumes in the receiving pool with undesirable foaming. In heat-exchanger equipment, the presence of the air plume can have a significant impact on the heat transfer characteristics. For all these applications the basic need is to determine the currents induced by the evolving gaseous phase in the surrounding liquid, and the subsequent mixing and partition of energy or species concentration in the flow.

A large amount of experimental work has been devoted to study plunging jets. Non-invasive techniques, such as photography, light scattering, laser-Doppler anemometry (Ben-Yosef et al., 1975) were employed to investigate the characteristics of plunging liquid jet including air entrainment, phase velocity fields and size of entrained air bubbles. For instance, Chirichella et al. (2002) studied air entrainment due to a translating axisymmetric laminar water jet plunging into a quiescent pool of water, where several different air entrainment modes were investigated depending on some critical values. High-speed visualization was used to successfully
capture these entrainment regimes. On the other hand, invasive techniques such as electrical conductivity microprobes (Serizawa et al., 1975) and optical fibre probes (Yu and Kim, 1991) were also used to study the jet penetration depth and the air carried under. These experimental studies were complemented by various analytical contributions which provided useful correlations to predict global flow features such as the onset of air entrainment, the amount of entrained air, the characteristics of the resulting bubbly plume and mass transfer, and the radial mean velocity and void fraction. The reader can refer to the comprehensive review of Bin (1993). On the basis of various experimental campaigns, Chanson (1997) and Chanson et al. (2004) established empirical relations to predict the entrained air content distribution and plunging jet flow motion. These correlations have been proven to be more precise compared to many theories proposed hitherto. Still, the validity of these empirical laws to predict gas penetration and concentration are challenged by different measurements with different flow properties and configurations; one of which is the present case study.

Numerical predictions of plunging liquid jets are also abundant in the literature. Sene (1984) proposed a sub-grid air entrainment model to predict the dispersed bubbly flow created by a plunging jet. The model was implemented into an Eulerian–Eulerian, two-fluid model to predict the gas holdup distribution underneath the free surface at different penetration depths, but turned out to be inaccurate in the area close to the entrainment region, see Schmidtke and Lucas (2009). The same two-fluid model was employed by Bonetto and Lahey (1993) for a round liquid–liquid jet. The Eulerian non-homogeneous multiple size group (MUSIC) model implemented into CFX was employed by Krepper et al. (2008).

The separation phenomenon of small and large bubbles seems to be well described by the model, although the modelling of bubble fragmentation and coalescence is still debatable in this context. Richards et al. (1994) simulated an axisymmetric jet from its formation to the dispersion in liquid–liquid systems using the Volume-of-Fluid method (VOF) without a turbulence model (the approach cannot be considered as a Direct Numerical Simulation (DNS) though). Liovic and Lakehal (2007a) were the first to use Large-Eddy Simulation for two-phase jetting flows, and applied it to gas injection in a Boiling Water Reactor (BWR) depressurization water pool.

The present study reports on findings of a combined experimental and simulation investigation of the behaviour of a plunging water jet under different flow configurations. The measurement data include the flow map regime, the jet penetration depth, the mean Sauter diameter of entrained air bubbles, and other flow characteristics like interface instability and re-entrainment. The numerical modelling uses the multiphase mixture model approach and the interface tracking technique (level-set), resorting both to the standard k-ε to model turbulence to predict the jet penetration below the free surface, its deformation and the liquid velocity and air holdup below the surface. A comparison between the two multiphase models is conducted against the data and semi-empirical results.

2. Experiments

2.1. Experimental Set up

Fig. 1 shows a schematic of the experimental setup conducted at FZD Rossendorf, Germany. The measurements were performed in a 0.3 m x 0.3 m x 0.5 m water tank equipped with transparent acrylic walls for visualization purposes. The water level was kept constant at 0.28 m. Water was pumped out of the tank and re-injected through a smooth 6 mm inner diameter and 0.55 mm long steel pipe used as a nozzle to produce a vertical jet. A high speed camera was used to capture images of the impact of the jet on the water. For each experimental condition, a 10 s long sequence of images of the region below the surface was taken. Backlighting was used during the experiments in order to ensure a proper exposure at the required filming rate. Experiments were realized for different nozzle heights and volumetric flow rates. The water velocity $V_0$ at the nozzle exit ranges between 1 m/s and 3.5 m/s. Based on the nozzle diameter $d_n$, the corresponding Reynolds number $Re = \rho d_n V_0 \mu_0 / \rho_l$ (where $\rho_l$ is the water density and $\mu_0$ is its viscosity) ranges between 5000 and 20,000. Assuming a critical Reynolds number for pipes of 2300, the range of flow parameters employed ensures a fully developed pipe flow at the nozzle exit. The jet length before impact was varied between 1 cm and 20 cm. Assuming free fall of the water after leaving the nozzle, the jet impact velocity, $V_i$ is defined as

$$V_i = \sqrt{V_0^2 + 2gL_f}$$

where $L_f$ is the jet falling length from the nozzle tip to the liquid surface. According to Eq. (1), this yields impact velocities $V_i$ between 1 m/s and 4 m/s.

2.2. Measurements of bubble size distribution

Bubble sizes were estimated from single images by means of image processing through subsequent background subtraction, cell segmentation, bubble detection and bubble size calculation using a Hough transform (HT) based algorithm. The dynamic generalised Hough transform (DGHT) algorithm (Leavers, 1989) has been proven to be particularly useful when the object to be recognized and reconstructed is symmetric, such as circles and ellipses. For distorted images of the bubble, the DGHT algorithm can be used to reconstruct their 2D shape. Nevertheless, it was found that this algorithm could only be used confidently for experiments with a small number of bubbles in the plume. When the plume is denser, bubbles overlap, obstructing the background light and distinguishing between them becomes rather difficult and in this particular case only bubbles outside the plume in the region of rising bubbles are identified and measured.
2.3. Measurements of penetration depth

The penetration depth is the lowest point of plume at a certain instant. Because the shape of the plume is not completely stable, a time averaged penetration depth is used. Fig. 2 shows two representative plume snapshots for jets with an impact velocity of \( V_j = 2.2 \) m/s. The jet with the higher flow rate (higher \( V_0 \)) entrains more bubbles, but its penetration depth is smaller in the snapshot. However, due to the fluctuation of the plumes snapshots are not conclusive. Therefore, for the calculation of the penetration depth, two different approaches were used. Both of them were implemented using the Image Processing Toolbox of Matlab. The first one is an averaging algorithm, which involves several processing steps for each image, e.g. background subtraction, filtering, removal of the small air bubbles at the walls and thresholding. Thus 2000 thresholded images were recorded over 10 s with a frequency of 200 frames/s to obtain an average. In view of the fact that temporal resolution is quite high, a total of 4 runs were made for the same experimental conditions. The second algorithm calculates the histogram of the penetration depth for each experimental sequence and in this case the penetration depth with the highest repetition rate is chosen. Results obtained from both methods were compared and seem to be in good agreement with one another. The difference between the results delivered by these two methods was found not to exceed 5%, in fact between 2% and 5%.

Fig. 3 presents pictures of time averaged concentration measurements for the cases mentioned previously. Despite having the same impact velocity, it can be observed that the left plume is less populated with bubbles than the right one with, however, a larger penetration depth. This reflects the general trend observed in this experiment, that higher flow rates produce higher entrainment rates. The phenomenon could perhaps be physically explained as follows: increasing the nozzle jet velocity with a fully developed turbulent pipe flow hence with an appreciable turbulence content – for a given diameter – may trigger jet-interface instabilities, which when impacting on the water surface entrains more air bubbles beneath the surface. In turn, jet momentum is diffused by the presence of the bubbles, causing a reduction of its penetration depth. The jet-interface instability phenomenon, a function of initial turbulence content, details of the nozzle design and length of free-falling jet, is therefore a key element affecting its lateral spreading and diffusion and penetration depth. Alternatively, larger flow rates in combination with jet-interface instabilities could generate larger air bubbles which are more buoyant and predominantly localised just below the impact point, see Igushi et al. (1998), and hence escape rapidly the gravity effect leading thereafter to a shorter penetration.

Looking again at the snapshots depicted in Fig. 3, one can easily obtain a penetration depth by localising the lowest large bubbles. For time-averaged gas plume pictures this is more difficult, since the edges of the plumes are smeared out. A slight estimation error is thus expected and needs to be considered when comparing the data with the calculations.

2.4. Measurement of flow regime map and bubble size distribution

One of the most interesting aspects of the present experiment is the qualitative results obtained for air entrainment beneath the pool free surface. Fig. 4a refers to experimental findings and illustrates the variation of the jet impact velocity against its free fall length before impact and identifies the flow regime taking place. Each data point represents one experiment for a given jet length and a nozzle velocity, resulting in a calculated impact velocity according to Eq. (1). The observed entrainment regime is reflected by the symbol type and colour where three regimes were observed; see Schmidtke et al. (2009). In general, the amount of entrained gas increases with \( V_0 \) if the jet length is kept constant (vertical rows).

The entrainment regime is only a qualitative criterion for characterizing bubble entrainment; quantitative criteria can be

![Fig. 2. Two different plumes for jet impact velocity \( V_j = 2.2 \) m/s.](attachment:image.png)
obtained by measuring the structure of the bubble plume and by detecting bubble sizes. Both can be obtained by an evaluation of the video images. It is observed from the regime map plotted in Fig. 4 that air entrainment by a plunging jet takes place when the jet impact velocity exceeds a critical value, which depends among other things on jet instability and its interaction with turbulence, see Cummings and Chanson (1997a,b). The Sauter mean diameter in the region of the rising bubbles is about 3–4 mm and is in agreement with previous data from the literature, e.g. Bin et al. (1996). The smallest observed bubbles are spherical with diameters between 0.3 mm and 1.5 mm (smaller bubbles are not resolved by the present measuring technique). The largest bubbles have horizontal diameters around the value of 7 mm. They appear however, only at high jet flow rates (high \( V_0 \)).

3. Numerical models

3.1. The mixture approach

The first approach employed to predict the two-phase flow pattern of the plunging liquid jet is the mixture model, see Manninen et al. (1996). The unsteady equations of continuity, momentum for the mixture and the volume fraction of a single dispersed phase are expressed as follows:

\[
\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}_m) = 0
\]  

\[
\frac{\partial \rho_m \mathbf{u}_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}_m \mathbf{u}_m) = -\nabla \rho_m + \rho_m \mathbf{g} - \nabla \cdot [\rho_m \mathbf{c}_p \mathbf{c}_g] + \nabla \cdot \mathbf{T}_m
\]

\[
\frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g \mathbf{u}_m - D_{km} \nabla \rho_g) = -\nabla [\rho_g (1 - \mathbf{c}_p) \mathbf{u}_g]  
\]

The mixture density \( \rho_m \) and the mixture velocity \( \mathbf{u}_m \) are defined as

\[
\rho_m = \sum_{k=1}^{k=2} \xi_k \rho_k
\]

where \( k \) denotes the phase \( k \), \( \mathbf{c}_p \) is the mass fraction of the dispersed phase.

In Eq. (3) and Eq. (4), the phase slip terms are expressed explicitly as functions of the relative velocity \( \mathbf{u}_{\rho g} \), which is obtained from the constitutive equation as follows:

\[
[\mathbf{u}_{\rho g}] \mathbf{u}_{\rho g} = \frac{4 \rho_c}{3 C_D} \left( \frac{\rho_g - \rho_m}{\rho_c} \right) \left[ \mathbf{g} - (\nabla \mathbf{u}_m) \mathbf{u}_m - \frac{\partial \mathbf{u}_m}{\partial t} \right]
\]

Note that \( p \) represents the dispersed phase and \( c \) the continuous phase. \( D_{km} \) is the diffusivity coefficient of the dispersed phase.

\[
\tau_{cm} = (\mu_m + \mu_{cm}) \left[ \nabla \mathbf{u}_m + (\nabla \mathbf{u}_m)^T \right] - 2 \rho_m k_m \mathbf{I}
\]

where \( k_m \) is the turbulent kinetic energy for the mixture, \( \mu_m \) is the dynamic viscosity for the mixture and \( \mu_{cm} \) is the turbulent eddy viscosity. \( \mathbf{I} \) is the unit tensor. The correlation of generalised stress tensor \( \tau_{cm} \) is written in the Reynolds Averaged Navier–Stokes (RANS) single-fluid or mixture context, representing both viscous and turbulent diffusion mechanisms. For the present work, it is assumed that the dispersed phase consists of spherical particles of a single average size \( \bar{d}_p = 2 \text{ mm} \). This particular value necessary for the calculation of interphase drag indeed contrasts with the measured mean Sauter diameter, which is only limited to certain regions of the flow particularly for dense jets. This is obviously a rough approximation and introduces an uncertainty. This is indeed one of the weaknesses of such phase averaged models. Inherent in the model is the assumption that the two phases move at different velocities, but do reach local equilibrium over a very short spatial length scale. Slip velocity function employed the algebraic slip method of Manninen et al. (1996). The inter-phase mass transfer does not apply here, and surface tension is neglected. The drag coefficient \( C_f \) appearing in Eq. (7) is determined by the correlation proposed by Schiller and Nauman, see Clift et al. (1978). The standard \( k-c \) two-equation turbulence model was used and all turbulent
quantities are based on the primary phase flow. Turbulence generation in the gaseous phase is not accounted for, and the turbulence of the primary phase is not directly affected by the presence of secondary phase. The viscous and diffusion stresses can often be omitted because the turbulence viscosity of the continuous fluid is much larger than the dynamic viscosity. These are two other assumptions of the model.

3.2. The level-set approach

The level-set technique (Sussman et al., 1994; Lakehal et al., 2002) is an Interface Tracking Method (ITM), where the interface separating the two phases is tracked directly in an Eulerian way, without phase averaging. In ITMs the flow system involves one single velocity–pressure field, with material properties including density, viscosity, and conductivity changing in time based on the phase marker scalar function. In the level-set technique the interface between immiscible fluids is represented by a continuous function $\phi$, representing the distance to the interface, which is set to zero on the interface and is positive on one side and negative on the other. The fluid motion equations under isothermal incompressible flow conditions without phase change or mass transfer expressed within this context take the form:

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial}{\partial t} \mathbf{u} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \rho \mathbf{g} + \nabla \cdot \mathbf{\tau} - \gamma \sigma \mathbf{n} \delta(\phi)$$

The last term on the right hand side of Eq. (10) denotes the surface tension force, with $\mathbf{n}$ standing for the normal vector to the interface, $\sigma$ for the interface curvature, $\gamma$ for the surface tension coefficient of the fluid, and $\delta$ for a smoothed Dirac delta function centred at the interface. The evolution equation for the level-set function reads:

$$\frac{\partial}{\partial t} \phi + \mathbf{u} \cdot \nabla \phi = 0$$

Material properties are updated locally based on $\phi$ and distributed at the interface using a smooth Heaviside function denoted by $H(\phi)$:

$$\rho, \mu, \ldots = \rho, \mu, \ldots |_L \cdot H(\phi) + \rho, \mu, \ldots |_G \cdot (1 - H(\phi))$$

where the subscripts $L$ and $G$ refer to liquid and gas, respectively. Depending on the flow, the level-set function ceases to be a signed distance from the interface after an advection step of Eq. (11). Various methods have been proposed to restore the level-set function correct distribution near the interface. In the code TransAT of AS-COMP used here, a re-distancing equation is solved, using the non-oscillatory 3rd-order WENO scheme, enhanced by a global mass-conservation scheme (Lakehal et al., 2002).

3.3. Computational parameters

Simulations have been carried out based on the experimental setup; the flow configuration is one in which a cylindrical tank is the receiving bath with the water level 280 mm above the tank bottom. The employed grid is generated to represent the injection...
pipe installed vertically above the free water surface and aligned with the axis of the tank (see Fig. 5). The diameter of the tank is 150 mm and the height from the nozzle exit to the water level was varied in the simulations to study the influence of jet length on the penetration and entrained-air concentration. In both simulations, a 2D axisymmetric domain was set covering the pool, the jet and the surrounding air. The structured mesh generated for the mixture-model simulations covered between 75,775 and 223,260 elements. The grids in the jet falling area (from the tube exit to the free surface of water) and the liquid part of the computational domain where jet penetration occurs were refined to track the accurate jet behaviour. For the level-set simulation, structured single-block meshes of 31,750–47,500 cells were used, consisting of one block only, albeit refined around the jet impact region and the free surface. The boundary conditions included an axis of symmetry, wall type, velocity inlet, and pressure and velocity.

The jet is injected through the nozzle with a constant bulk velocity. A pressure outlet condition was adopted at the top of the domain fixing the pressure to atmospheric conditions. To maintain a constant water level, an outlet condition was used in the bottom-right corner of the tank for the mixture-model simulations. The mass flow rate of water leaving the tank was set equal to the water mass flow rate introduced from the nozzle. In the level-set simulations, a pressure outlet was also imposed at the right boundary above the tank, so that water in excess could leave it and compensates the excess. It was assumed that the flow in the core of the nozzle in the flow upstream is fully developed, so that the turbulence intensity at inlet can be estimated from the following empirical correlation for pipe flows: $I = 0.16 \left( \frac{Re}{C_0} \right)^{-1/8}$, where $Re$ is the Reynolds number based on the nozzle diameter. However, the backflow turbulence intensity for the pressure outlet was estimated according to previous experiences and it was set to be 0.05%.

The equations in the homogeneous model simulations are discretized using the 2nd-order Upwind scheme within a finite-volume formulation. The PISO algorithm was used for pressure–velocity coupling. An adaptive time-marching strategy was adopted during the calculation. Computations were continued till the global residuals reached the absolute criteria set in that code: $10^{-4}$ for continuity, velocity and volume fraction, and $10^{-3}$ for $k$ and $\epsilon$. The time for calculation convergence depends on the case studied and ranges from 0.5 s to 3.5 s. The code TransAT used for level-set simulations is finite-volume based too, using the SIMPLEx algorithm for pressure–velocity coupling. Time advection is achieved using the 3rd-order Runge–Kutta explicit scheme. The convective fluxes are discretized using the 3rd-order quick scheme. The $k-\epsilon$ model is employed for turbulence, with specific near-interface treatment (Liovic and Lakehal, 2007b). The surface tension coefficient is set equal to 0.072 N/m. Transient simulations were averaged over time between 1 and 1.7 s which is probably not long enough to obtain statistically averaged results.

4. Results and discussions

Both experimental and numerical results are presented. Experimental findings will serve to validate and complement the calculation results. The present computations target the continuous regime of Fig. 4. Fig. 4b represents the corresponding calculated target regime for both multiphase models used which exhibit an excellent agreement with experiments. The results are discussed in terms of flow regimes, characteristics of entrained bubbles, jet penetration depth and falling water jet deformation for which experimental results and visualisations are available. The air-water flow properties such as the liquid velocity and the air hold up under the free surface are also discussed. The discussion will also contrast the performance of the mixture and level-set approaches.

4.1. Jet penetration depth

Fig. 6 compares the jet penetration depth for nozzle exit velocities of 1.8 m/s and 2 m/s, respectively. The penetration depth is plotted against the jet velocities at impingement given by Eq. (1). Fig. 6 includes the results from the empirical relationship of Bin (1984), where the penetration depth $h_p$ is determined by:

$$h_p = 2.11 \frac{V_j}{d_0}$$

The results of penetration depth from the experiments are obtained using the averaging algorithm mentioned in Section 2. In the mixture model calculations, the penetration depth is defined as the location where air volume fraction reaches zero at the bubble plume edge. In the level-set simulations, this criteria is somewhat correlated to the vertical speed of the jet, which should converge to zero.

From Fig. 6, it can be seen that the simulation results and the experimental values are relatively overall in good agreement. The behaviour is well captured and the results reveal that both the mixture model and the Interface tracking variant are able to capture the plunging liquid jet penetration depth. Bin’s expression is also plotted in the figure and as expected, it displays a fundamentally different behaviour. This can be explained by the fact that Bin’s correlation is based on global parameters like jet impact velocity and diameter of injection nozzle, but neglects jet falling/breaking length and jet-free surface interaction, which are important parameters in this phenomenon and have considerable influence on the jet penetration depth. It also ignores viscous shear at the interface and surface tension. Fig. 6 shows, however, different results between the two modelling approaches: while the predictions are equally good for low impact velocities, the level-set is relatively better than the mixture model for high impact velocities, for nozzle exit velocities of 1.8 m/s and 2 m/s. Several reasons could be invoked to explain the discrepancies, among which the fact that the level-set predicts a wide range of frequency scales or unsteadiness of the flow than the average model. The mixture model tends naturally to smear the interfaces, which could lead to wrong definition of the topology of the flow, and thus
under-estimates the shear-induced turbulence. Be it as it may, it is too premature to attribute these drawbacks to turbulence modelling alone.

Fig. 7 compares the volume fraction contours predicted by both models, for jet nozzle velocity of 1.8 m/s. The penetration location is also indicated on the figures. The simulation results for the impact velocities plotted in Fig. 6 are shown here (three values only for level-set simulations). It is obvious that both simulations require more samples to achieve complete statistical convergence. The right panels depict a smeared interface, which is an artefact of time averaging only; the interface is actually sharper in the level-set simulations as indicated further below in Fig. 8. The \( L_j = 12 \) cm and 16 cm calculations in the left panels show incomplete statistical convergence, which should have predicted a complete ascension of the gas swarm to the free surface. Differences between the homogeneous model and level-set are clearly highlighted when looking at the \( L_j = 2.5 \) cm results.

### 4.2. Jet instability and free surface deformation

As the water jet penetrates ambient air, a gaseous film develops along its trajectory as an envelope surrounding its surface. The formation of a gas film adjacent to the surface of the jet is associated with the relative velocity between gas and liquid phases, resulting in the generation of surface jet instabilities and deformations. According to previous studies, both axisymmetric and asymmetric deformations can occur on the jet surface depending on its length, nozzle exit velocity – and perhaps the pipe diameter and surface tension as well, as illustrated by the images from the experiment of plunging liquid jets provided by Schmidtke et al. (2009). A detailed theoretical analysis of jet instability is beyond the scope of this paper; we restrict ourselves to reporting the simulations results and how they compare with the data.

Fig. 8 depicts the simulation (using both models) and experimental topologies of the jet interface around the water-bath free surface, right at impingement on the water surface. It can be seen that the mixture-model simulations fail to capture the shape of the jet interface with the present 2D modelling assumptions and computational parameters, in particular the Rayleigh type of instability depicted in the left experimental panel and the bubble entrainment at the free surface. The level-set simulations reveal the existence of the above mentioned instability in the left panel, depicting a stronger entrainment and more violent free surface deformation in the right panel. The reasons for these discrepancies are the same as evoked previously. A 3D modelling strategy is perhaps necessary to capture such details, using a finer mesh. The videos produced during the measurement campaign suggest that the bubbles trapped at the surface partly originate in the core liquid subsequent to jetting effects and mixing and then rise by buoyancy effects, and partly induced by jet breaking at the impingement entraining air. The videos show that these bubbles trapped at the surface could remain ‘floating’ for a very long time (seconds). This is obviously computationally unaffordable.

### 4.3. Velocity and air hold up below the free surface

Below the impingement point, the flow structure basically comprises two distinct regions: (1) a diffusion cone with a downward flow motion induced by the plunging liquid jet, and (2) a swarm of rising bubbles which surrounds the former one (Chanson, 1997). In the present study no measurements of liquid velocities and air concentrations were performed. However, in this section the computed distribution of liquid velocity and void fraction using the mixture and level-set methods are discussed and compared with semi-analytical and empirical results taken from the literature. The velocity distribution in the fully developed region valid beyond a distance of 10 jet diameters from the origin of the jet for a round turbulent single phase jet is given by the following semi-analytical correlation, see Chanson (2004):

\[
V_j = \frac{5.745}{x/d_0} \cdot \left( \frac{1}{1 + 0.125(18.5 \varphi)} \right)
\]

where \( x \) is measured from the plane of impingement of the plunging jet. The profile predicted by Eq. (14) together with the computational results from the mixture model and the level-set variant are plotted in Fig. 9. The general shape of the velocity profile is Gaussian as found by McKeogh and Ervine (1981), Bonetto and Lahey (1993), Hammad (2010), and Ma et al. (2010). It can be seen that the velocity of the entraining liquid column decays laterally from the nozzle centreline to reach zero away from the jet centreline. The source of the discrepancies between the simulations and theoretical profiles can be traced to the nature of Eq. (14) which, strictly speaking, is valid for liquid–liquid jet flow, while the velocity distributions of impinging jet flow will be affected by the drag/buoyancy of the entraining gas. Nevertheless, the profiles generated...
by the two models are consistent in trend with the profile of Eq. (14), although the two models predicted lower velocities away from the centreline than Eq. (14). A further observation indicates that the level-set model predicts marginally higher diffusion, by its radial spreading, in contrast to the mixture model. The maximum velocity value of the jet on the centreline is somewhat over predicted by both models. The local minimum predicted by the mixture model and its subsequent shape is another deficiency, in that they reflect a higher concentration of air around the jet head. The bubbles are not allowed to rise up to the free surface, a drawback rooted in the slip model used, as already suggested by Fig. 7. A consistent explanation of the phenomenon in support of this analysis is given below when discussing void fraction distributions.

The distributions of mean void fraction obtained by the mixture and level-set methods are displayed in Fig. 10. For the discussed case, the penetration depth predicted by the mixture model is 14.5 cm while for level-set it is 15.5 cm. Thus in order to probe the air concentration distribution around this area, the planes with

$$h = 12\text{ cm}, 13\text{ cm} \text{ and } 14\text{ cm} \text{ are considered for the purpose. In addition to the numerical results, air concentration values obtained from the empirical correlation proposed by Chanson et al. (2004) are also shown:}$$

$$C = \frac{Q_{air}}{Q_w} \left( 1 + \frac{r}{r_{max}} \right)^2 \times I_0 \left( \frac{r}{2D^*} \right)$$

where $$Q_{air}$$ is the air flux, $$Q_w$$ is the water jet flow rate, $$x$$ is the longitudinal coordinate, $$D^* = 2D/(V_d)$$ is the dimensionless air bubble diffusivity, $$D_1$$ is the advection diffusion coefficient, which averages the effects of turbulent dispersion and streamwise velocity gradient, and $$I_0$$ is the modified Bessel function of the first kind of order zero. To keep the empirical nature of Eq. (15), the values of $$Q_{air}/Q_w$$ and $$D^*$$ were determined from the best fit of data from the bank of experiments performed by Chanson et al. (2004). They are assumed

![Experimental snapshot](image1)

![Simulation](image2)

![Level-Set snapshot](image3)

**Fig. 8.** Jet instability and surface deformations at impact (snapshots). Comparisons between the models and the data for case (a) $$V_0 = 1.2\text{ m/s}, L_j = 25\text{ cm}$$, and case (b) $$V_0 = 3\text{ m/s}, L_j = 25\text{ cm}$$.
constant in the radial direction. Further, $Y_{c_{\max}}$ denotes the lateral distance from the centreline when the air content reaches the maximum, which is probably the weakest point in the above expression.

The choice of $Y_{c_{\max}}$ is important in the calculation as it decides the radial location of the air content peak value. The predicted peak value of air concentration differs when either the mixture approach or the level-set is used. Therefore two different $Y_{c_{\max}}$ values are assigned for the mixture and level-set models respectively.

Fig. 9. Empirical versus computational radial average velocity profiles at $h = 5$ cm, 6 cm and 7 cm ($V_j = 2.12$ m/s, $L_j = 2.5$ cm).

Fig. 10. Empirical versus computational radial gas profiles at $h = 12$ cm, 13 cm and 14 cm ($V_j = 2.12$ m/s, $L_j = 2.5$ cm).
when comparing to the theory. The air content peak location in the level-set simulations is 0.3 cm for all three positions, while for the mixture model; this location is predicted at 0.9 cm, 1 cm and 1.1 cm, i.e. at around 1 ± 0.1 cm, which is around three times larger than the level-set results, reflecting an over estimation of the rate of lateral spreading. Comparing the mixture model and the level-set results, it is found that there is an over-prediction of the maximum concentration values by both the level-set and mixture models. Overall, the prediction of the mixture model is better than the level-set model at $h = 14$ cm, whereas the reverse is true at $h = 12$ cm, with slightly better prediction by the level-set than the mixture model of the maximum concentration at $h = 13$ cm. The lateral diffusion of the jet is considerably higher with the mixture model than with level-set and the rate of decay of air concentration predicted by the mixture model is higher than the one predicted by the level-set. The level-set results seem to be more consistent with the empirical relation, in the light of the previous comparison of velocity profiles. Fig. 9 has indeed shown that at the centre of the jet within the pool, the maximum average velocities are higher when predicted by the mixture model than both the level-set and the theory. This is synonymous to a higher air concentration in the jet centre line beneath the free surface, which is clearly seen in the context of Fig. 10 comparing air void fractions. This confirms our observations made earlier that the mixture model is not capable of predicting the rise of air bubbles to the free surface. It is seen that the trend of the distribution of air content is globally similar between CFD and the empirical relation. This gives rather additional credibility to Chanson’s correlation, albeit it leaves the air content peak value as an unknown of the model.

5. Conclusions

The present work reports findings from a combined experimental and simulation campaign centring around a plunging turbulent water jet on a free surface of a water pool. The CFD approaches employed for the purpose are deliberately different; the mixture model and level-set approach, (which belongs to the interface tracking framework). The methods were applied under two-dimensional axisymmetric conditions to predict several of the flow characteristics. Attention was paid to jet penetration depth and jet liquid column instability and free surface deformations. The jet penetration depths obtained from the two computational approaches are in good quantitative agreement with the data. Jet instability and surface deformations are predicted, but not with the same accuracy: Interface Tracking Methods are better suited to mimic the details of this flow and somehow show the trends occurring in the experiments, but are computationally more expensive. 2D axisymmetric simulations are not sufficiently precise to replicate the flow details which are 3D in nature, but are clearly capable of returning the macroscopic features of the flow.

The liquid average velocity field and air concentration of the entraining two phases under the water pool free surface were also numerically investigated and compared to existing analytical correlations. The results show reasonable agreement between simulations and empirical correlations, in the case of the liquid velocity decay from the centreline, but noticeable differences were revealed for maximum velocity at the jet centreline and associated bubble concentration. The mixture model predicts a higher velocity than the level-set and the theory at the early stage of jet penetration, due to a higher concentration of air that cannot rise to the surface; a model drawback buried in the way the slip velocity is algebraically modelled. As for the air concentration distributions, the two methods display a similar distribution map. However, the location of the maximum air content and the peak value of air holdup are different in quantity. This makes the comparison with the theory rather difficult since the correlation considered here includes precisely the air content peak value as a model unknown. In conclusion, the two numerical techniques can generally grasp the global characteristics of plunging liquid jets and the differences in results lie in the different algorithms and computational time for the simulations.

Acknowledgements

The work was conducted within; the EC-funded projects NUR-ISP and THINS, in the framework of the 7th EURATOM Frame work Program.

References
