Bubble clouds generated by droplets impacting water

Name of the graduate program (Master 1) :
Laboratory for the internship : LMFA
Principal Investigator : Jean-Philippe MATAS (jean-philippe.martas@univ-lyon1)

Summary: A liquid jet or stream of droplets impacting a liquid surface and thereby, entraining the surrounding gas to form a bubble cloud is among the most-common daily-life phenomena. Understanding and modeling such bubble clouds is crucial for multi-phase mixing processes in nature and in industrial techniques for pollution control, turbine operation and performance, oxygenation of water resources, among others. From a more fundamental point of view, the physics of such "bubbly" flows is at the margin of classical fluid dynamics. Based on recent studies in our group which provide a strong theoretical basis and large amount of experimental data, there is wide scope to further improve our understanding via direct numerical simulations of such two-phase flows. In this context, the internship aims to characterize the bubble cloud formed by circular jets, and droplet trains, using numerical simulations (Basilisk Flow Solver).

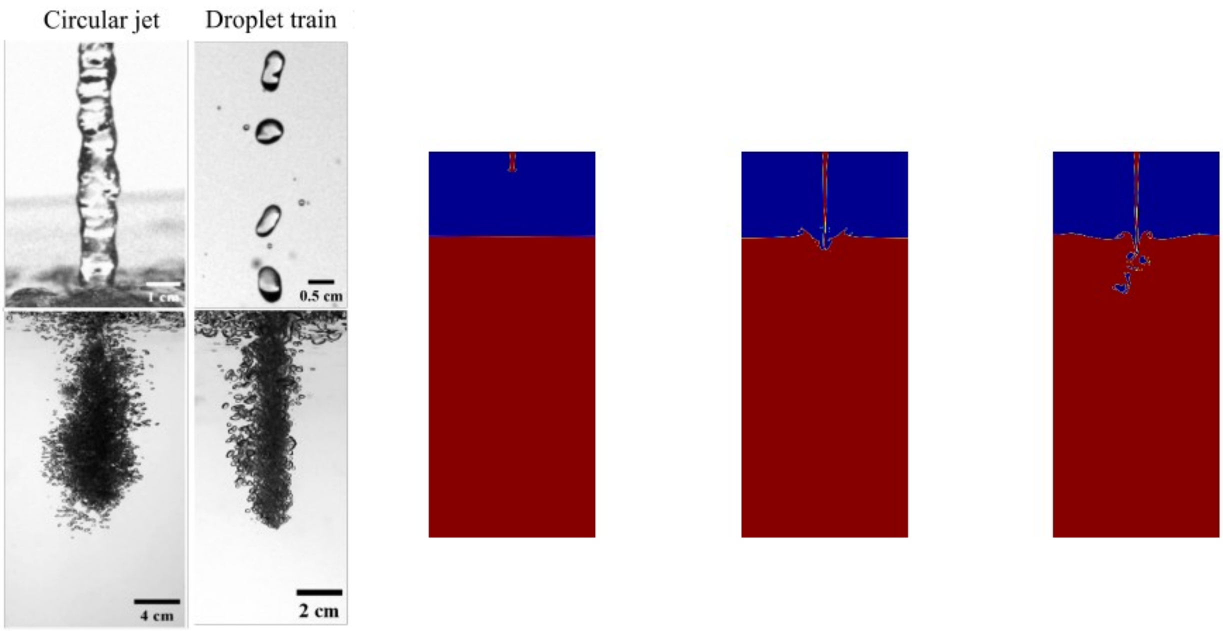

Figure 1: A stream of jet/droplets impacting on water (left: experiments; right: an example of VOF simulations)

Scientific description of the project: The impact of a liquid jet on a reservoir entrains air, forming a two-phase flow, known as bubble cloud. Such aerated jets and the air entrained after impact are also valuable to ensure optimal oxygenation of reservoirs or rivers, contributing to the sustainability of the aquatic ecosystem, as in the project 'Petit Saut'(EDF - French Guiana). To gain proper control of the air entrained and its properties (e.g. bubble size distribution, velocity), it is necessary to relate the properties of the jet during impact to the parameters that influence the penetration of air under the surface. In an on-going project, namely, JetPlume (funded by ANR), with regards to hydro-electric power stations of EDF[†], major safety challenges include robust prediction and risk control of erosion downstream of dams. It implies understanding the water flow evolution which influences downstream civil engineering structures, such as dissipation basins, concrete and rough rock. One of the main concerns is, therefore, to control the vector of these

[††] Électricité de France

impacts, namely the plunging jets and their subsequent dynamics. A thorough understanding of the evolution during free fall, impact against a liquid surface and consequences of plunging jets (i.e., generated bubbly flow within the reservoir) is then key to advance our capabilities of modeling these flows and to enable efficient, safe and environmentally friendly energy production.

While previous studies focused on single jets, real-world scenarios often involve large jets breaking into multiple jets and droplets, as seen with waterfalls. Thus, here we study bubble clouds formed by a series of droplets using Direct Numerical Simulations. Recently, a powerful predictive model was developed [1] to scale bubble cloud size from micro-jets to huge waterfalls. The model is based simply on the momentum balance between the freely-falling jet and the two-phase mixing layer. Our latest findings [2, 3] shows that the model gives excellent prediction of cloud depth formed by circular jets using the measured values of fraction of air (void fraction) inside the cloud.

As for numerical simulations, these flows are extremely challenging due to the large gas/liquid density ratio and the associated high Reynolds numbers. Their main characteristic is the presence of an air/water interface with a thickness that can be neglected compared to other flow scales. One of the main ingredient in the current theoretical models is the addition of buoyancy forces in the momentum balance as the bubbly jet extends into the liquid reservoir after impact. Using high-tech optical probes which can measure bubble sizes and speeds, a huge amount of data is available from experiments with circular plunging jets [4]. However, hardly any numerical simulations had been attempted for cases where the gas/liquid fraction is large after impact. This is the motivation and the context of the present internship project. Basilisk Flow Solver [5] is an open-source for Direct Numerical Simulation of fluid flows. It has been successfully used to make robust simulations of complex two-phase flows. Also, we have used it to study hydrodynamic instabilities of entrainment flows in the presence of strong inertia. Firstly, we wish to characterize the bubble cloud depth formed by circular jets, and droplet trains, using 2D simulations in Basilisk Flow Solver. Numerical results will be validated by comparison with existing small-scale laboratory experiments. Then, droplet size and frequency will be modified to analyze their role on the cloud depth. Further investigations on the air entrainment processes will be obtained by analyzing bubble-laden just below impact. The time evolution of air cavities and the dynamics of cavity collapse will be studied. Also, we hope to obtain new insights to model the large air/water void fraction in experiments.

References:

[1] Guyot, G., Cartellier, A., & Matas, J. P. (2020). Penetration depth of a plunging jet: from microjets to cascades. *Physical Review Letters*, 124(19), 194503.

[2] Dev, N., Jerome, J. J. S., Scolan, H., & Matas, J. P. (2024). Liquid inertia versus bubble cloud buoyancy in circular plunging jet experiments. *Journal of Fluid Mechanics*, 978, A23.

[3] Dev, N., Scolan, H., Jerome, J. J. S. & Matas, J. P. (2025, under revision). Bubble Clouds Formed by Multi-Plunging Jets. *International Journal of Multiphase Flow*

[4] Dev, N. Bubble Clouds Formed by Jets and droplets. *Thèse UCBL*, 2022-2025

[5] Popinet*, S.* Basilisk*. http://basilisk.fr/.*