

## DEVELOPMENT OF A SUB-GRID MODEL FOR SELF-AERATION

P. LOPES<sup>1,2</sup>, J. LEANDRO<sup>1,2</sup>, R. F. CARVALHO<sup>1,2</sup>, G. TABOR<sup>3</sup>

<sup>1</sup>MARE – Marine and Environmental Sciences Centre, University of Coimbra, Portugal

<sup>2</sup>DEC – Department of Civil Engineering, University of Coimbra, Portugal

<sup>3</sup>CEMPS – College of Engineering, Mathematical and Physical Sciences, University of Exeter, UK

*pmlopes@student.dec.uc.pt; leandro@dec.uc.pt; ritalmfc@dec.uc.pt; g.r.tabor@exeter.ac.uk*

**Keywords:** *OpenFOAM, Self-aeration, Sub-grid model, Volume-of-Fluid*

The numerical simulation of flow self-aeration processes is viewed as one of the most ambitious subjects under current investigation. Engineering applications of this topic are vast, for instance the simulation of river oxygenation in environmental engineering, estimation of energy losses occurring in a spillway dam for design project, or even to predict the loss of efficiency in urban drainage elements due to air in pipes. The main challenge resides in the simultaneous simulation of the largest scales of free-stream flow along with the interface between water and air (order of scale from metres to kilometres), together with the smallest scales of interface fluctuations (order of millimetres). The smallest scales are likely to be smaller than the grid size, but are still important in the simulation of bubble formation, their transport and iterations through the flow. This process also needs to be done without significant increases in the computational time.

To deal with this we have developed a unified framework solver to simulate the natural entrainment of air using the Volume-of-Fluid (VOF) methodology. It combines the well-stabilised two-fluid solver *interFoam* to predict the location of the free-surface, with a standard transport equation to treat the air as a pseudo-single-phase (Figure 1). The air, naturally entrained at the free-surface is calculated along the interface area, based on its turbulent energy and surface fluctuations and inserted into the transport equation as a source term. As this term relies on the solution of a thin interfacial region of well-defined thickness, the interface cannot be diffusive and mesh independence tests are imperative [1].

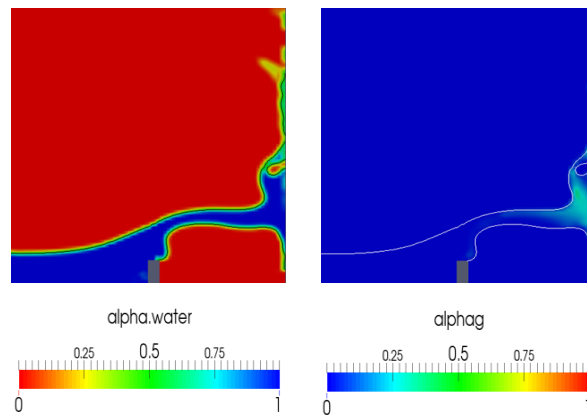
At this workshop we intend to demonstrate the applicability of this solver in some engineering cases as well as the reliability of the source term working at the interface.

### Acknowledgements

This study was funded by FCT (Portuguese Foundation for Science and Technology) through the Projects UID/MAR/04292/2013 and Grant SFRH/BD/85783/2012, financed by MEC (Portuguese Ministry of Education and Science) and FSE (European Social Fund), under the programs POPH/QREN (Human Potential Operational Programme from National Strategic Reference Framework) and POC (Human Capital Operational Programme) from Portugal2020.

### References

- [1] P. Lopes, G. Tabor, R. F. Carvalho, and J. Leandro, “Modelling air-entrainment in circular, 3D plunging jet using the OpenFOAM CFD toolbox,” in *Advances in Numerical Modelling of Hydrodynamics workshop*, 2015.



**Figure 1: 2D dam break case: at left: pure VOF simulation; at right: air entrained due to impact of water on right wall. The variable <math>\langle\text{alphag}></math> stands for air fraction. Free-surface delimited by <math>\langle\text{alpha.water}>=0.5</math>.**