34th Danubia-Adria Symposium on Advances in Experimental Mechanics
University of Trieste, Italy, 2017

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF WATER JETS

Freddy Florez[1], Reinhard Prenner[1] and Norbert Krouzecky[1]

[1] Vienna University of Technology, Institute of Hydraulic Engineering and Water Resources Management, Karlsplatz 13/222 1040 Vienna, Austria. E-mail: norbert.krouzecky@tuwien.ac.at

1. Introduction

The determination of the breakup length in free falling water jets is difficult, because it is a complex process where the interaction of independent variables as the level of turbulence, the oscillating pressure, nozzle geometry and ambient properties influence the flow characteristics of the jet [1].

In order to improve understanding of air entrainment into water, a so-called free falling water jet ejecting downwards into quiescent air from a circular nozzle has been tested in the laboratory and the results have been validated by means of two-phase Computational Fluid Dynamic (CFD) models.

2. Experimental rig

Experimental tests were carried out in the Hydraulics Laboratory of the Institute of Hydraulic Engineering and Water Resources at the Vienna University of Technology. The model rig facility included a 5 m high steel trestle in order to generate free falling water jets into the atmosphere through a nozzle directly attached to the pump line (DN 150 mm) of the laboratory as is schematized in Fig. 1. The model rig is described in detail in [1] and [2].

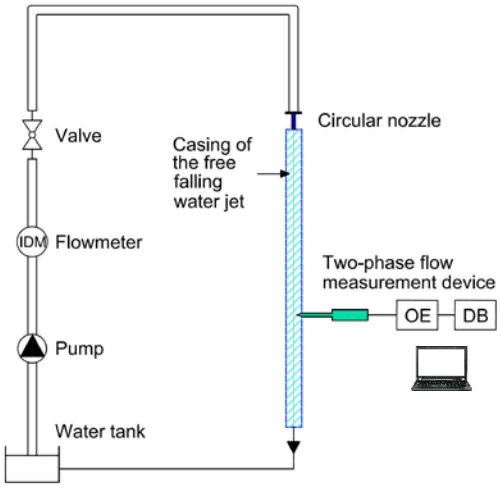

Fig. 1. Experimental rig.

Once the water jet left the nozzle, it fell vertically within a jet casing box to the floor of the tailrace channel to the water tank.

2.1 Nozzle

The nozzle was composed of a circular cylinder built in PMMA and a flange of PVC on one side in order to attach it to the steel flange of the water supply pipe (see Fig. 1). The nozzle had a diameter d = 0.02 m, an exit length L_n = 0.35 m and the sharp edge (90°) of the nozzle inlet was smoothed into a circular arc of diameter $d_n = 4 \times 10^{-3}$ m.

2.2 Instrumentation

The discharge Q_w was regulated manually by a valve located in the feeding pipe and it was measured by an inductive flowmeter (IDM).

A sapphire fibre-optical double probe manufactured by RBI Instrumentations was used to measure the local air concentration C (%) at several cross sections from the nozzle exit. The two-phase flow measurement device comprised the following elements (see Fig. 1): a probe with a double sapphire tip, an opto-electronic amplification module (OE), a data acquisition board (DB), a PC and the software ISO-Lite for data processing and analyzing.

3. Computational Fluid Dynamics

The open source software OpenFOAM v. 2.3.1 was selected as the tool to carry out three-dimensional (3D) numerical calculations for solving the Navier-Stokes equations for two incompressible fluids, capturing the interface using the Volume of Fluid (VOF) method [3]. Two types of turbulence modelling were implemented: 1) standard k-ε model, which is a type of Reynolds averaged simulation (RAS) and, 2) k-equation eddy-viscosity model based on Large Eddy Simulation (LES) models. The computational fluid dynamic domain is shown in Fig. 2, where d_2 = 0.06 m and 0.1 m for RAS and LES models, respectively. The nozzle outlet velocity U_0 = 7.96 m/s generated an atomization mode of disintegration, confirming the expected behavior according to the Ohnesorge diagram.

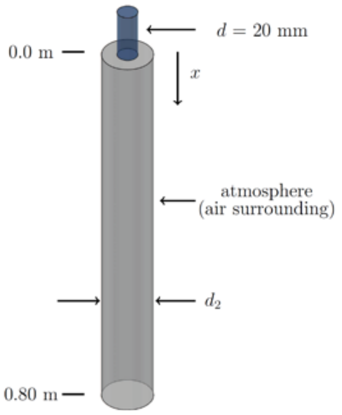

Fig. 2. Computational fluid dynamic geometry.

The minimum computed cell sizes during the construction of the mesh are summarized in Tab. 1.

Model used	Δx [m]	Δy [m]	Δz [m]	Number of cells
RAS *k*-ε model	2×10^{-3}		2×10^{-3}	0.65×10^{6}
LES *k*-Eqn	$0.65\ 10^{-3}$		1×10^{-3}	2.31×10^{6}

Tab. 1. Cell size characteristics.

4. Results

In Fig. 3 are shown the jet disintegration results for the experimental test (left), the CFD simulation by using the standard *k*-ε turbulence RAS model (center), and the CFD simulation by using the k-equation eddy-viscosity turbulence LES model (right) are shown.

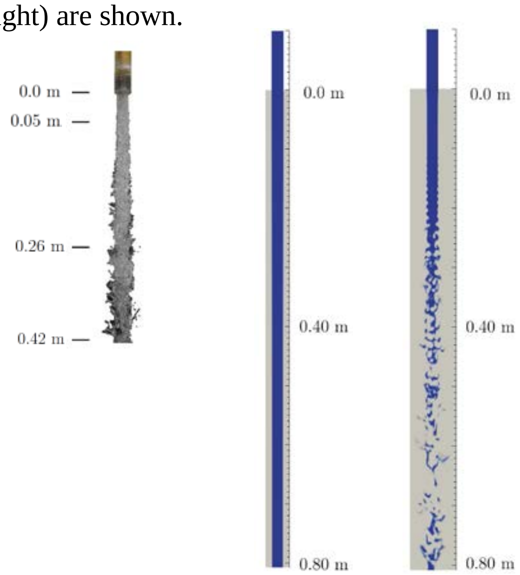

Fig. 3. Experimental test (left), CFD *k*-ε turbulence RAS model after 2.0 s (center) and, CFD k-equation eddy-viscosity turbulence LES model after 2.0 s (right).

In Fig. 4 the air concentration distributions of the experimental test (Exp) and the numerical simulations by using the two turbulence models mentioned at a distance of 0.05 m and 0.683 m from the nozzle exit are presented. The relative position *y/r* is defined as the distance *y* from the jet axis and *r* is the radius of the jet.

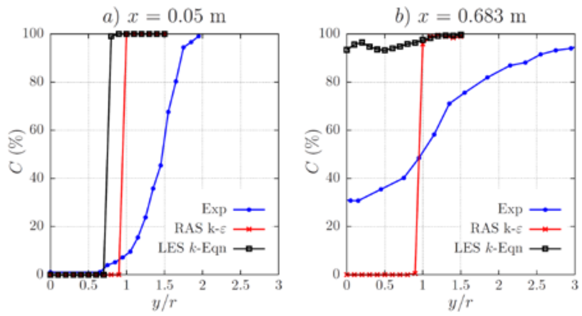

Fig. 4. Air concentration measured and computed at several locations.

5. Remarks

- The standard *k*-ε turbulence RAS model in neither case generated water separation from the surface and no breakup took place. Likewise, the angle of spray was always negative due to the acceleration of gravity.
- The k-equation eddy-viscosity turbulence LES model generated an early breakup and it was able to reproduce the dynamic interaction between the phases air and water. Despite this model requiring a computational time twice that of the RAS model, its use is suggested in hydraulic engineering for studying problems in which the air is of concern.

Acknowledgements

To Vienna Scientific Cluster (VSC) Research Center for his support.

References

[1] Florez, F., Air Entrainment in Free Falling Water Jets, PhD Thesis, TU-Wien, 2016.

[2] Florez, F., Prenner, R., Krouzecky, N., Measurements of Air Concentration and Velocities in a Free Falling Water Jet, Transactions of FAMENA, 40, 2016, pp. 57-68.

[3] Greenshields, C., OpenFOAM User Guide version 4.0, The OpenFOAM Foundation, 2016.