EXPERIMENTAL AND NUMERICAL CHARACTERISATION OF A JET IMPINGEMENT ON A POOL

RITA F.CARVALHO\(^{(1)}\), M.NAZMUL A.BEG\(^{(1)}\) & PEDRO M.LOPES\(^{(1,2)}\)

\(^{(1)}\) Marine and Environmental Research Center - MARE, Dep. Civil Engineering, University of Coimbra, Coimbra, Portugal.
\(\text{e-mail:} \text{ritalmfc@dec.uc.pt, mnbeg@uc.pt, pedromiglopes@gmail.com}\)

ABSTRACT

Flow features on a pool subjected to an impingement jet were studied experimentally and numerically. Data from experimental measurements and numerical simulations were compared with some available data from literature. Experimental data used Acoustic Doppler Velocimetry (ADV), which is a widely used tool for the characterization of fluid flow and turbulence as well as image analysis to understand numerical and experimental results. The numerical model was applied within OpenFOAM® toolbox utilizing Volume of Fluid method. Model allows the study of the influence of several jet factors, such as jet fall height in the pool hydrodynamics. Characteristics of free-surface perturbation, velocity, pressure, air concentration and turbulence field of pool are presented and discussed.

**Keywords:** Jet, Plunge pool, ADV, image analysis, OpenFOAM®.

1 INTRODUCTION

The air-entrainment along the jet is caused by the process of jet disintegration due to turbulence and secondary interactions with the surrounding atmosphere. The high velocity impact on a pool causes vorticity and air entrainment to be produced. The relationship between turbulence (destabilizing force), surface tension and gravitational forces (stabilising forces) and their effects on roughness and free surface breaking are particularly relevant. Air entrainment produces a set of bubbles, with the large ones being affected by buoyancy with a flow reorientation towards the free surface. According to Ervine's experimental evidence (Ervine, 2004), the air-entrainment into jets impacting a plunge pool depends on the size of the disturbances of the free surface generated by the impact. The effect of pre-aeration on the jet characteristics, as well as tangential effects of jets at different viscosities are discussed in Kiger and Duncan (2012). Miwa et al. (2018) presented a synthesis of experimental works done in vertical jets into a pool and created models to predict air-entrainment, based mainly on Weber number. Recent findings on jet velocities, turbulence structures and air concentration profiles in flat jets can be found in Bertola et al. (2018). Studies of non-vertical jets impacting pools are restricted. Some examples are the studies of Castilho et al. (2014), who focused on a jet in a real dam, using CFD, Deshpande et al. (2012) who studied a horizontal jet and its impact on the plunge pool with an experimental and computational study, and Lian et al. (2018) who studied the influence of sediment concentration on the characteristics of a horizontal jet at a certain height of the surface pool.

It is concluded that the simulation of the impact of the jet is not simulated with the necessary rigor. It seems that, at low tilt angles, the air intake near the impact is less random than in jets with more vertical impact. In particular, when the jet reaches the free surface, the velocities are dominated by the concentration of the momentum, generating ejection of fluid, that impacts more downstream in a weaker way. The characteristics of the jet and its impact on a pool with different inclinations are not known.

In this work, we performed an experimental work to characterize velocity field in a plunge pool where a horizontal jet from a certain height falls toward the plunge pool. We performed CFD, using OpenFOAM of the experimental set-up, using interFoam and also for different horizontal jets falling from different heights. Details of the experimental and numerical method are presented in section 2 and 3, respectively. At the experimental work, we used Acoustic Doppler Velocimeter, which is a versatile, high-precision instrument that measures all three flow instantaneous velocity components. Acoustic Doppler velocimeters (ADVs) are capable of reporting a good description of flow turbulence when certain conditions are satisfied (Voulgaris & Trowbridge, 1998; García, et al., 2005). However when air-entrainment is relevant, data is distorted. Spikes in the water velocity signals recorded using ADVs may be caused by phase shift ambiguities, aeration effects and high turbulent intensities (Romagnoli et al. 2013). Presence of spikes, Doppler noise and filtering effects due to the ADVs' sampling strategy could strongly affect the turbulence characterization based on the recorded signals (Romagnoli et al., 2013). A Go-Pro camera was also used to catch jet impinging behaviour.
2 EXPERIMENTAL WORK

The experimental work was performed in an open channel flume with both acrylic bottom and sidewalls to facilitate optical access installed at the Laboratory of Hydraulics and Environment of University of Coimbra (UC). Figure 1 shows the schematic diagram of the jet flow into a plunge pool at the flume, 1.00 m wide, 1.20 m deep and 12.00 m long, where the water height was set to 0.25 m. The water jet was set through 18.5 mm polyethylene pipeline with inlet velocity \( (U_i) \) of 4.6 m/s, in a closed circuit using a centrifugal pump, pumping from a constant head reservoir, fed by the pool. The resulting key non-dimensional numbers for this configuration are Reynolds number, Weber number and Froude number (Deshpande et al., 2011), being \( \rho = 1000 \text{ kg/m}, \mu = 1 \times 10^{-3} \text{ Kg/m/s}, \sigma = 0.072 \text{ N/m}, \) and \( g = 9.81 \text{ m/s}^2 \) is given by:

\[
Re = \frac{\rho DU_i}{\mu} = 8.5 \times 10^4, \quad We = \frac{\rho DU_i^2}{\sigma} = 5.4 \times 10^3, \quad Fr = \frac{U_i}{\sqrt{gH}} = 11.98 \quad (\text{as } U_i = \sqrt{U_i^2 + 2gH} = 5.1 \text{ m/s})
\]

Figure 1 also shows the Vectrino arrangement to measure the velocity in the area limited by dashed lines with various distances from the wall. This parallelepiped measurement area consists in a half flume since symmetrical conditions were observed, with 0.35 m wide, 0.24 m deep and 0.50 m long containing different planes with several distance from the wall. The coordinate system defined for this experiment has its origin in the bottom right side of the flume from the sidewall, with the longitudinal x-axis parallel to the bottom and the flume axis, the y-axis transversal and with the z-axis perpendicular to the bottom. To characterize the flow field in this region, velocity measurements were carried out in these planes with 30 second recording length (3000 samples) for each point. ADV measurements were taken in a total of 560 points for all planes, with a maximum spacing of 0.05 to 0.1 m between points in the y- axis and 0.02 to 0.04m in the z- axis. The reference of the planes (0,0) was 20 cm from the wall of the flume. The Vectrino was installed in the flume and connected to the computer using Vectrino Plus software with specific configuration 100 Hz sampling rate, + or – 1.00 or 4.00 m/s Nominal velocity range, 1.8 mm transmit length, 4 mm sampling volume, high power level and XYZ coordinate system.

![Figure 1. Schematic diagram of the jet impinging on a plunge pool at the Laboratory of Hydraulics and Environment of University of Coimbra (UC).](image)

Before each measurement signal-to-noise ratio (SNR), signal amplitude and correlation were checked to ensure data quality and the velocity range was set to span the entire range of measured velocities to avoid phase wrapping. According to the distance of each plane, all ADV measurements had SNR values between 50 - 80%. Data was post processed for despiking and noise reduction using winADV. Although noise and spikes can be reduced or eliminated in many cases by adjustment of probe operational parameters, there are some situations in which spikes cannot be entirely avoided. In the jet impingement area, ADV data was not useful, since a lot of air and turbulence was present, decreasing the data quality and the adequate filtering techniques. A complete signal analysis was carried out using winADV to identify and remove the influence of errors, like presence of spikes, Doppler noise and filtering effects due to the ADVs’ sampling strategy, on the computation of turbulence parameters. After this procedure, the velocity values were evaluated and imported to MATLAB for the analysis of average and fluctuations.

Pictures from experimental set were used to understand air-entrainment, experimental and numerical results. Figure 2 presents two pictures, which give an idea of jet impingement characteristics.
3 NUMERICAL WORK

3.1 CFD Codes and Equations

OpenFOAM, is a widely used open source library which includes different solvers to solve partial derivative equations or group of them, using different numerical schemes. CFD code used is based on the Navier-Stokes equations / Reynolds-Averaged Navier-Stokes equations governing the motion of the 3D incompressible and isothermal flows in which the free surface is described using a Volume-Of-Fluid (VOF). According to this description, the VOF-function $F = F(x; y; z; t)$, ranging from 0 to 1 corresponding respectively to cells without water and full occupied by water, is included in the mass and momentum conservation equations and it is updated using an advection equation for $F$ (Hirt and Nichols, 1981, Carvalho et al., 2008). Further in the mass and momentum conservation equations, VOF-function is included through physical properties as density and viscosity which are defined by a weighting of the values for air and water.

Some improvements of VOF models were developed including surface tension and the interface curvature as well as artificial compression of the interface to improve accuracy of the interface (Weller, 2008). The VOF method used in the interFoam solver implemented in the OpenFOAM (Equations 1 to 3) has two particularities: a volumetric surface force, explicitly estimated by the Continuum Surface Force (CSF) function of the surface.

Figure 2. Downstream underwater view (top) and Side view (bottom) images of the plunging jet showing the formation of bubbles and the splashing produced by the impingement process (UC).
tension, and the interface curvature, which are included in the momentum equation (Brackbill et al., 1991); the compression of the interface that is achieved by introducing an extra, artificial compression term in the advection equation, the third term of Eq. [3] (Rusche, 2002).

\[ \nabla \cdot \vec{u} = 0 \]  
[1]

\[ \frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p^* - g \cdot \vec{x} \rho + \nabla \cdot \tau + f \]  
[2]

\[ \frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{u}) + \nabla \cdot \left[ \frac{\vec{n}_\alpha (1 - \alpha)}{C_\alpha} \right] = 0 \]  
[3]

where \( \vec{u} \) is the mean velocity vector, \( p^* \) is the modified pressure adapted by removing the hydrostatic pressure from the total pressure, \( \alpha \) is the VOF function, \( t \) is the time, \( \rho \) is the fluid density, \( g \) is the acceleration due to gravity, \( \tau \) is the shear stress tensor, \( f \) is the volumetric surface tension force (where CSF and interface curvature are included) and \( \vec{u}_c \) is the compression velocity, that acts as a velocity perpendicular to the interface and is written as:

\[ \vec{u}_c = C_\alpha |\vec{u}| \frac{\nabla \alpha}{|\nabla \alpha|} \]  
[4]

where \( C_\alpha \) is a coefficient that activates \( (C_\alpha = 1) \) or disables \( (C_\alpha = 0) \) the interface compression term. The factor \( \nabla \alpha / |\nabla \alpha| \) returns the normal to the interface calculated on the gradient of \( \alpha \), and it is used to orient the magnitude of the velocity \( |\vec{u}| \) (Lopes et al. 2017).

For high volume fractions, the volume of entrained air must be taken into account in the calculation of physical properties of the mixture (two-way coupling). Physical properties, \( \varphi \), such as \( \rho \) or \( \mu \), are weighted on the fluid fraction present in each cell:

\[ \varphi = \alpha_i \varphi_i + \alpha_g \varphi_g \]  
[5]

We used a SST \( k - \omega \) turbulence model. This is known for the best combination of two Reynolds-Averaged Simulation (RAS) formulations, using high-Reynolds-number formulation of \( k - \varepsilon \) model for the free-stream region and taking advantage of the accuracy and robustness of \( k - \omega \) model in the near-wall zone.

3.2 Numerical Schemes

Total Variation Diminishing (TVD) limited form of central differencing is used for convective terms in momentum equation. The Van-Leer scheme is used for the convective term in VOF-advection equation and ‘Interface Compression’ scheme is used in order to bound the solution of the compressive term between 0 and 1. To ensure boundedness of the phase fraction and avoid interface smearing, the solution of the VOF equation is done with the Multidimensional Universal Limiter for Explicit Solutions (MULES). The Pressure-Implicit with Splitting of Operators (PISO) procedure proposed by Issa (1986) is used for pressure velocity coupling in transient calculations with 3 loops. We used the same discretization schemes employed in previous works on the jet tested with positive outcome (Lopes et al., 2016).

3.3 Model Construction

The 3D computational domain represents the flume, with water and air and the pipe, through which the jet is created (see Fig. 1). The geometry was constructed in SALOME-9.2.2, and the different generated stl were used to define boundaries and to construct the mesh in cfmeshv1.1 (Juretić 2015). The total number of cells in the domain, mainly hexahedra cubes, was 8.6 M being the interior mesh size chosen as 20 mm with minimum \( \Delta x = 0.006 \) m, next to the pipe. Figure 3 shows the computational mesh for a jet height of \( z=0.5 \) (\( H =0.25 \) m, measured from water surface until jet axe).

As initial conditions, we considered a water depth of 0.25 m with velocity zero in the flume and the pipe full of water with a velocity of 4.6 m/s, as verified in experiments. The upstream top of the pipe was the inlet, which was kept as constant velocity. The flume transversal section below the inlet was divided in two: considering water part as the outlet with velocity calculated to have the same flow rate as the inlet to maintain water volume in the domain; and the upper part as wall. The opposite transversal section, the bottom and side glasses were considered walls as well as the pipe inside, outside and borders. The upper part of the domain was atmosphere.

The models were run for 10 seconds using 32 cores at MPI mode. The cluster computing system at the University of Coimbra was used for this part. The central plan XZ was analysed to get main parameters of the jet impinging, such as impact length, L, and angle, \( \theta \), as well as dimension of pool perturbation.
Figure 3. Mesh used for plunging jet numerical study.

4 RESULTS AND DISCUSSION

Figure 4 presents comparisons of ADV data, numerical model results obtained with interFoam and the side view image, which serves for interpretation of the results.

Figure 4. Plunge pool x-z central plane with selection areas: 0.5 x 0.25 m in the impingement area: a) image of experiments; b) ADV mean velocity magnitude and streamlines; c) Velocity field obtained by numerical simulations.
Figure 4 includes selected areas, central area in dashed point, where ADV data must be rejected, as well as an area, continuous line ellipse, indicating possible vortices. As said, the area of the jet impingement contains strong turbulence with air, which produces spikes. After filtering techniques, only a small amount of values, stands, giving very small values.

Results of the simulations utilizing current solver for RANS equations with VOF method, for the jet presented in Figure 5 and Table 1. Figure 5 presents the main parameters of the jet falling from z=0.5 m, the height of 0.25 m from the pool free-surface (H) and bending under the influence of gravity, impacting the free surface at different distances (L) and at different angles with respect to the free surface (θ), as well as different plans XZ and XY for the tested jet (H=0.25) to have an idea of the perturbation area in the pool. Reynolds and Weber numbers are approximated 8.5 x 10^4 and 5.4 x 10^3. Froude number was around 11, since Impact velocity (U_i) is 5.1.

![Diagram](Image)

Figure 5. Definition of Impinging Parameters and Plunge pool general aspect at different plans at the pool (XY z=0, 0.05, 0.1, 0.15, 0.2 and 0.25; and XZ y=0; 0.1; 0.2; 0.3; 0.4 and 0.5): a) z=0.5; H=0.25 m;

Table 1. Parameters for the jet characterized by D = 18.5 mm, U_0 = 4.6 m/s, Re=8.5 x 10^4, We= 5.4 x 10^3 and Fr =11

<table>
<thead>
<tr>
<th>H (m)</th>
<th>H/D (-)</th>
<th>L/D (-)</th>
<th>θ (°)</th>
<th>U_i (m/s)</th>
<th>F_r (-)</th>
<th>X/D</th>
<th>Y/D</th>
<th>Z/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>13.51</td>
<td>45.95</td>
<td>25</td>
<td>5.11</td>
<td>11.98</td>
<td>47.30</td>
<td>25.34</td>
<td>16.22</td>
</tr>
<tr>
<td>0.04*</td>
<td>1.00</td>
<td>10.0</td>
<td>12.5</td>
<td>4.10</td>
<td>6.54</td>
<td>21.5</td>
<td>-</td>
<td>5.0</td>
</tr>
</tbody>
</table>

*Deshpande et al. (2011)
5 CONCLUSIONS
If the present study, a horizontal jet reaching a plunge pool was studied numerically and was tested in the laboratory, where some photos and some measures with Vectrino were analysed. Images gave a good idea of perturbation area in the pool and help to interpret results. Vectrino measures in the impinging zone data was discarded since a lot of air and turbulence induced spikes, which leaded to very small values of the velocity. However, they were important to identify downstream vortice.

The simulations with InterFoam doesn’t allow to simulate the jet with the necessary rigor including air-entrainment. However, the jet can be estimated computationally with enough accuracy to evaluate main parameters, such as the characteristics of free-surface and perturbation in the pool hydrodynamics, due to several jet factors, as the jet fall height and flow discharges.

ACKNOWLEDGEMENTS
This study had the support of the FCT (Portuguese Foundation for Science and Technology), through the Project UID/MAR/04292/2019, which was financed by MEC (Portuguese Ministry of Education and Science) and the FSE (European Social Fund), under the programs POPH/QREN (Human Potential Operational Programme from National Strategic Reference Framework) and POCH (Human Capital Operational Programme) from Portugal 2020.

All the numerical results here showed were performed on the Centaurus Cluster of the Laboratory for Advanced Computing of University of Coimbra, Portugal. All the ADV measurements was done by Heslam Alrayess, PhD Student, Department of Civil Engineering, Faculty of Engineering, Ondokuz Mayis University, Samsun, Turkey, during his Erasmus period at University of Coimbra.

REFERENCES

