E-proceedings of the 36th IAHR World Congress
28 June – 3 July, 2015, The Hague, the Netherlands

Experimental and numerical analysis of velocities in plunge pools of free falling spillways

LUIS G. CASTILLO[(1)] & JOSÉ M. CARRILLO[(2)]

[(1)] *Universidad Politécnica de Cartagena, Cartagena, Spain,*
luis.castillo@upct.es

[(2)] *Universidad Politécnica de Cartagena, Cartagena, Spain,*
jose.carrillo@upct.es

ABSTRACT

Turbulence constitutes one of the principal mechanisms of energy dissipation in plunge pools. In falling jets and in dissipation basins appear high turbulence and aeration phenomena that cannot be correctly studied by the classical methodologies.

Given this problem, and based on the different experimental results obtained in the study of free falling jets, researchers are characterizing such flows in the Laboratory of the Universidad Politécnica de Cartagena (Spain). Velocities, pressures and air entrainment rates are being obtained and analyzed.

The methodology of computational fluid dynamics (CFD) can simulate the interaction between different fluids, such as the two-phase water-air flows case. The CFD codes are based on the numerical solution of the Reynolds-averaged Navier–Stokes equations (RANS), together with turbulence models with different degrees of complexity. In this way, the results obtained in laboratory are compared with the complementary numerical model.

Keywords: energy dissipation, falling jets, numerical simulations, overtopping, plunge pool

1. INTRODUCTION

In recent years, the increasing magnitude of design floods has prompted re-evaluations of spillway capacity and operational scenarios for large dams throughout the world. Current capacity of many spillways is inadequate, raising the possibility that dams might be overtopped during extreme events. This creates new loading scenarios for the dam and raises questions about erosion and scour downstream from the dam (Wahl *et al.*, 2008).

Nappe flow constitutes one of the types of plunge pool operation in the overtopping of dams. In turbulent flow, pressure fluctuations are the main mechanism affecting the incipient movement of the particles. The erodibility index relates the relative magnitude of the erosive capacity of water and relative resistance of the material (natural or artificial) to resist erosion (Annandale, 1995, 2006).

Different empirical formulae may be used to characterize the hydrodynamic actions in plunge pools. Due to the great difficulty of analyzing prototypes, all of the them have been obtained by using diverse experimental facilities and reduced scale models.

Energy dissipation of the jet may be divided into four steps: disintegration of the jet during the falling (aeration and atomizing), air entrainment when the jet enters into the plunge pool, diffusion in the plunge pool and finally impact with the bottom of the plunge pool.

Modeling of falling jets is difficult because the break-up and air entrainment characteristics of the jet are influenced by both surface tension and turbulence effects. In addition to this, the hydrodynamic actions are also affected by the turbulence scale. In a physical model scale effects will appear. However, their effects may be minimized or accounted for through careful choice of the model size and careful interpretation of the results.

Two conditions are defined to consider the overtopping design:

- The issuance conditions, located at the outlet of the spillway. This section of the jet corresponds to $z = -h$, where z is the vertical component with origin on the weir crest, and h the energy hear over the weir crest.
- Impingement conditions, obtained in the cross section of the jet just before impact with the plunge pool.

In the issuance conditions, the mean flow velocity $V_i = (4gh)^{1/2}$ is considered. In the impingement conditions, situated at entrance of the jet into the pool, we can consider the mean velocity, V_j, and the impingement jet thickness, $B_j = B_g + 2\xi$, in where B_g is the thickness by gravity conditions (the jet thickness decreases due to gravity effect), and ξ is the jet lateral spread distance by turbulence effect and is approximately equal to the square root of the fall distance (Davies, 1972).

Another important parameter is the jet break-up length, L_b. Beyond this distance the jet is completely developed (disintegrated). It no longer contains a core but consists essentially of blobs of water that disintegrate into finer and finer drops. For flows smaller than 0.25 m^2/s (laboratory tests values), the Horeni´s formulae $L_b \sim 6q^{0.32}$ (cited by Ervine *et al.*, 1997) seems to be correct (Castillo, 2006).

From the study of free falling rectangular jets, Castillo (2006, 2007) proposed a parametric methodology for calculating the impingement jet thickness B_j, reviewed by Carrillo (2014) and Castillo *et al.* (2014b):

$$B_j = B_g + 2\xi = \frac{q_j}{\sqrt{2gH}} + 4K_\varphi T_u \sqrt{h}\left(\sqrt{2H} - 2\sqrt{h}\right) \qquad [1]$$

where q_j is the specific flow in the impingement condition, H the drop height, $T_u = \overline{V_i'}/V_i$ the turbulence intensity in the issuance condition. $\overline{V_i'}$ and V_i are the root mean square and mean jet velocity in the main direction, and $K_\varphi = \overline{V_i'}/w' \approx 1.24$, where w' is the vertical component of the turbulent velocity.

In falling jets and dissipation basins it is difficult to carry out studies based only on classical methodologies. The computational fluid dynamics (CFD) programs allow researchers and designers to evaluate different effects with a smaller cost than that incurred building scale models. There are studies modeling spillways with which produces accurate results. However, the study of overflow nappe impingement jets has not been sufficiently examined. Simulations of free air-water overflow weirs are scarce, and require small mesh sizes and a high computational effort.

Turbulence in the falling jet has been analyzed using computational fluid dynamics techniques. Results obtained with CFD are compared with laboratory measurements and empirical formulae. To identify the level of reliability of computed parameters, validation of air entrainment and velocity along free falling jets, thickness and break-up of jets, and pressures on the bottom of the plunge pool, are carried out by using a two-fluid model, turbulence models and mesh-size analysis.

This work analyzes and compares the velocity profiles measured in the plunge pool, with numerical simulations.

2. EXPERIMENTAL FACILITY

The hydraulics laboratory at the Universidad Politécnica de Cartagena in Spain has a turbulent jet experimental facility in which the energy dissipation of turbulent rectangular jets is being studied. The mobile mechanism allows researchers to vary the discharge heights between 1.70 and 4.00 m and flows from 10 to 150 l/s. It has an inlet channel with a length of 4 m and width of 0.95 m, in which different dissipation systems have been located. The weir is a sharp crest with a height of 0.37 m.

The plunge pool, in which different water cushions may be regulated, is a 1.60 m high and 1.05 m wide box made of methacrylate. Instantaneous pressure measurements were registered with piezoresistive transducers located on the plunge pool bottom, kinetic energy at the inlet channel with Acoustic Doppler Velocimeter (ADV) equipment, mean velocities and air entrainment rate in different sections of the falling jet with optical fibre instrumentation.

ADVs have become highly useful in fluid dynamics and are applied to the study of three-dimensional flow and turbulence in both the laboratory and field (rivers, channels and hydraulic structures, amongst others).

The flow was measured with a V-notch weir, located downstream from the plunge pool. The discharge rate of the V-notch was tested with a velocity-area method using ADV equipment upstream from the weir. Differences between V-notch results and the velocity-area method were smaller than 5% of the current flow.

The setting characteristics of the ADV were selected considering that the main objective is to measure the mean velocity and macroscopic turbulence. In this way, the velocity range was selected as ±0.30 m/s with a frequency of 10 Hz, avoiding the noise generated by the equipment when higher frequencies are used. With this setting, the ADV equipment was able to measure the time-averaged flow field with an accuracy of better than ±0.002 m/s. The kinetic turbulence measured 0.50 m upstream the weir in the experimental facility was used as the inlet condition in the numerical simulations. The device was also used to measure the velocity profiles at different sections of the plunge pool (Figure 1).

3. NUMERICAL MODEL

For the turbulent flow, CFD codes solve the differential Reynolds-Averaged Navier-Stokes (RANS) equations of the phenomenon in the fluid domain, retaining the reference quantity in the three directions for each control volume identified.

In preparing this study, an extensive literature review of hydraulic dams was carried out. However, given that the CFD methodology is relatively recent there are few well documented references for free overflow spillways. For this reason, it is necessary to review CFD accuracy in similar typologies.

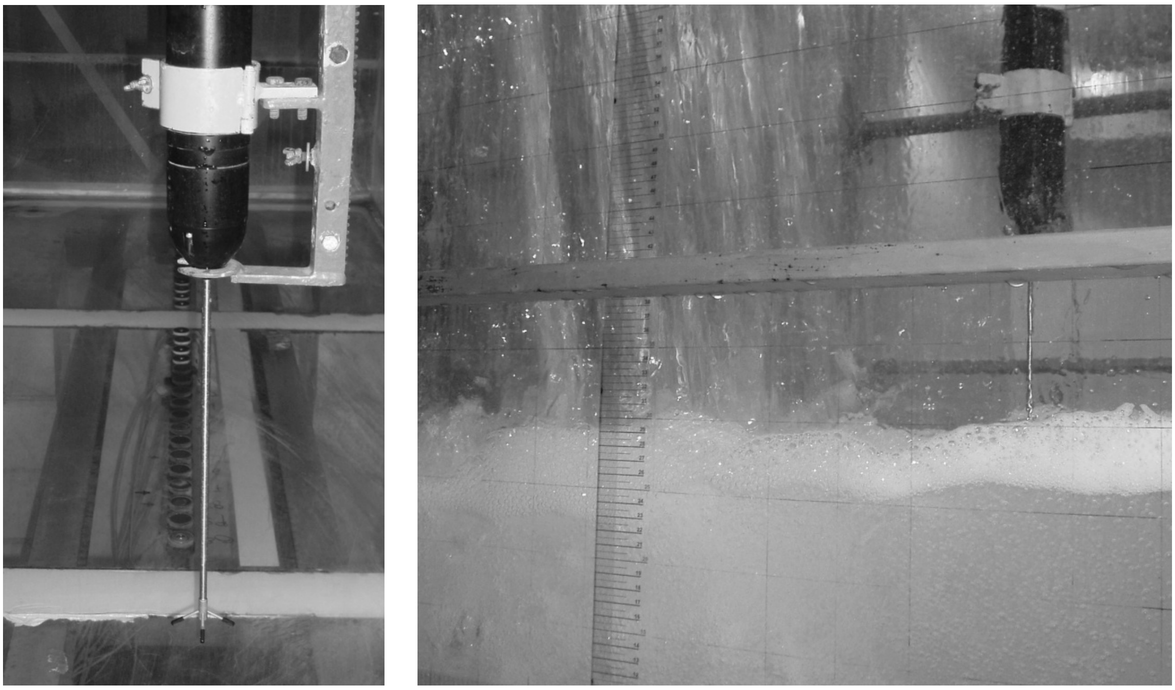

Figure 1. Acoustic Doppler Velocimeter in the plunge pool.

For the numerical modeling, the CFD volume finite scheme program ANSYS CFX (v. 14.0) has been used.

The fluid domain is divided into control volumes, which must satisfy the balance of the governing equations. The code allows different types of elements to be solved. The main difference between the types of elements is the number of nodes used to solve the equations within each control volume. A larger number of nodes per element obtains a more accurate solution in their internal resolution. Following Castillo *et al.* (2014a) and Carrillo (2014), the mesh size was 0.01 m based on hexahedral elements, approximately the half of the impingement jet thickness for the tests carried out.

All scenarios were obtained by a transient calculation time of 60 seconds, using 20 Hz frequency. The transient statistics were obtained by considering that permanent conditions are reached after 20 seconds of simulation (Figure 2).

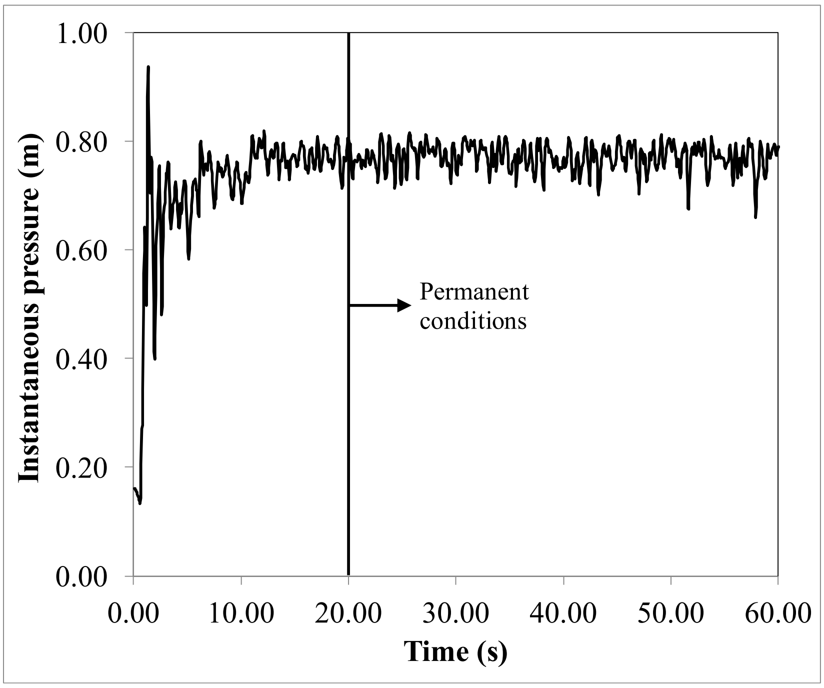

Figure 2. Transient of pressure at the stagnation point of the plunge pool.

In order to reach the closure of the Navier-Stoke equations, turbulence models can be used. There are different approximations, from one-equation turbulence models to the direct simulation.

As a compromise between accuracy and computational effort, the RANS turbulence models are widely used. Eddy-viscosity turbulence models consider that such turbulence consists of small eddies which are continuously forming and dissipating, and in which the Reynolds stresses are assumed to be proportional to mean velocity gradients.

Castillo *et al.* (2014a) and Carrillo (2014) tested different turbulence models in the falling jet case. In this work, the SST turbulence model has been selected. The SST model takes into account the accuracy of the *k-ω* model in the near-wall region and the free stream independence of the *k-ε* model in the outer part of the boundary layer. To do this, the original *k-ω* model (Wilcox, 2006) is multiplied by a blending function F_1, while the *k-ε* model (Launder and Sharma, 1972) is transformed to a *k-ω* formulation and multiplied by a function (1-F_1) (Menter, 1994). F_1 is designed to be one inside the boundary layer and decreases to a value of zero away from the surface.

Residuals are defined as the imbalance in each conservation equation following each iteration. The solution is said to have converged if the scaled residuals are smaller than prefixed values ranging between 10^{-3} and 10^{-6}. In this work, the residual values were set to 10^{-4} for all the variables.

To solve the air-water two-phase flow, the Eulerian-Eulerian multiphase flow homogeneous model was selected. In each control volume, the sum of the volume fraction of all phases is the unit.

In general, it may be assumed that the free surface is on the 0.5 air volume fraction. However, due to the high air entrainment in the nappe, the jet thickness and the break-up length were calculated using a 0.8 air volume fraction. Figure 3 shows the mesh size and the free surface obtained with the CFD program when permanent conditions are reached.

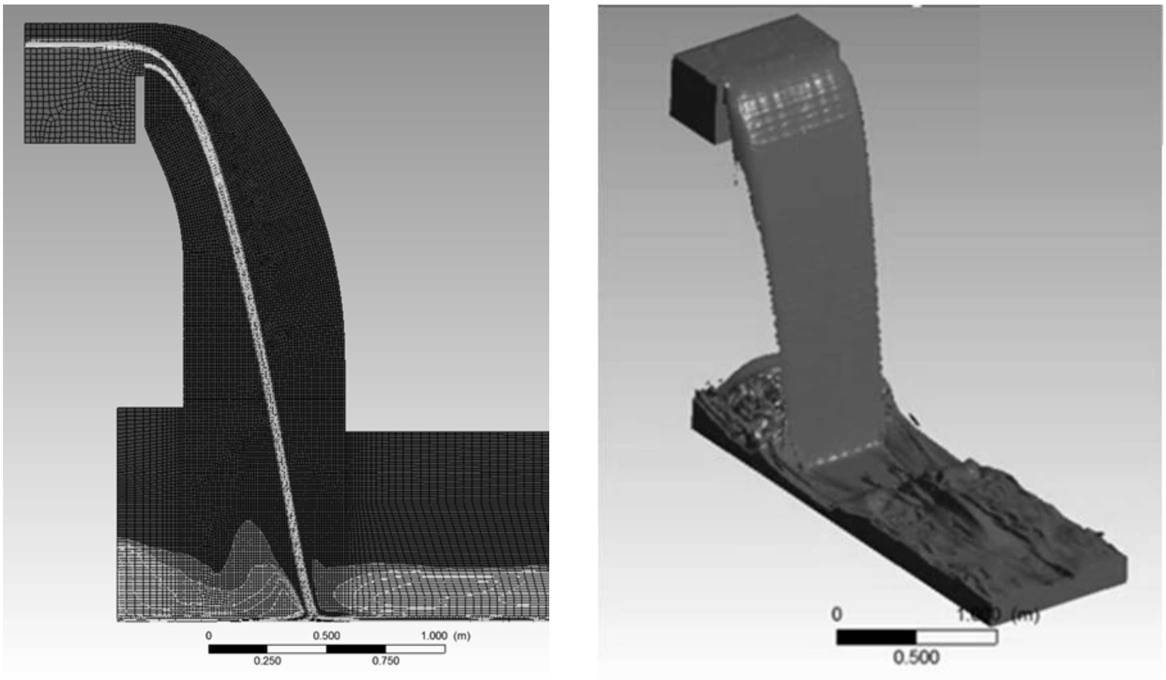

Figure 3. Mesh size and free surface of turbulent jet.

The model boundary conditions corresponded to the flow, the turbulence at the inlet condition obtained with ADV (located 0.50 m upstream of the weir), the upstream and downstream levels and their hydrostatic pressures distributions.

The inlet condition considers the mass flow rate with a normal direction to the boundary condition, the turbulent kinetic energy, and the water level height at upstream deposit. For simplicity, the symmetry condition in the longitudinal plane of the plunge pool was used.

The outlet condition has been considered as an opening condition with flow normal to the boundary condition and hydrostatic pressure. The water level height at outlet has been modified according to the water cushion depth, *Y*, in the laboratory device.

4. RESULTS AND DISCCUSION

Figure 4 shows the velocity vectors obtained with the CFD program near the stagnation point. The upstream and downstream flow recirculation regions in the vicinity of the impact of the falling jet, and the development of the bottom jet downstream the stagnation point may be observed.

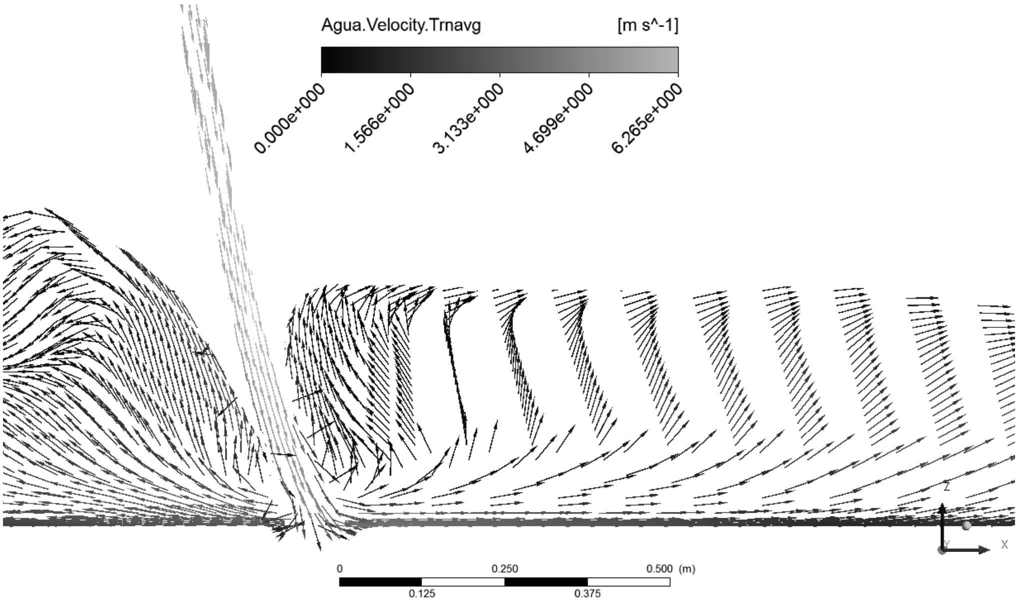

Figure 4. Mean velocity vectors simulated with CFD.

To analyze the velocity profiles downstream of the stagnation point, the plunge pool has been divided into diverse cross sections spaced 0.10 m. The numerical simulations and the laboratory measurements carried out with the ADV equipment have been compared. In Figure 5, the horizontal and vertical components of the mean velocity profiles have been analyzed, together with the turbulent kinetic energy in different sections of the plunge pool. The presence of a bottom jet seems to be clear. However, it could not be properly measured in the laboratory with the ADV equipment. This is due to its reduced thickness and the difficulty of to carry out measurements with Doppler equipment in highly aerated flows (see Figure 1). After certain distance, the velocity profiles tend to have an uniform distribution, with similar values in laboratory and in the simulations.

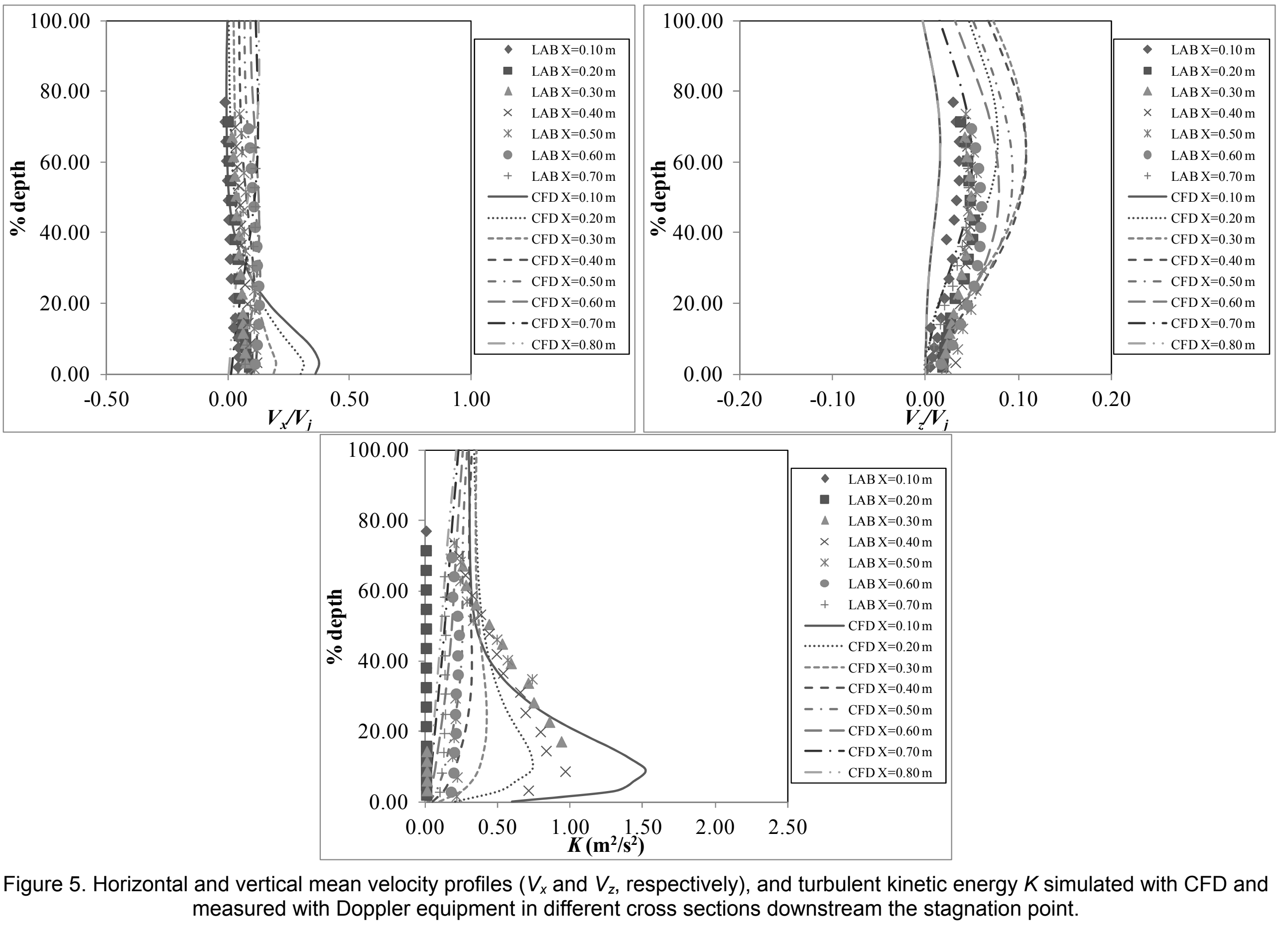

Figure 5. Horizontal and vertical mean velocity profiles (V_x and V_z, respectively), and turbulent kinetic energy K simulated with CFD and measured with Doppler equipment in different cross sections downstream the stagnation point.

To estimate the dimensionless mean horizontal velocity of the jet in the case of hydraulic jumps, diverse authors propose adjustment expressions (Table 1).

Table 1. Expressions for calculating dimensionless velocity profiles.

Author	Expression
Görtler (1942), cited by Liu *et al.* (1998)	$\frac{V}{V_{máx}} = 1 - tanh^2\left(0.881\frac{y}{\delta_l}\right)$
Rajaratnam (1976)	$\frac{V}{V_{máx}} = e^{-0.693\left(\frac{y}{\delta_l}\right)^2}$ $\frac{V}{V_{máx}} = 1.48\left(\frac{y}{\delta_1}\right)^{1/7}\left(1 - erf\left(0.68\frac{y}{\delta_l}\right)\right)$
Hager (1992), cited by Chanson and Brattberg (2000)	$\frac{V}{V_{máx}} = 2\left(5\frac{y}{\delta_l}e^{1-5\frac{y}{\delta_l}}\right)^{0.12}$
Ohtsu *et al.* (1990)	$\frac{V}{V_{máx}} = e^{-0.5\left(1.765\frac{y-\delta_{máx}}{\delta_l}\right)^2}$ $\frac{V}{V_{máx}} = e^{-0.5\left(1.814\frac{y-\delta_{máx}}{\delta_l}\right)^2}$
Lin *et al.* (2012)	$\frac{V}{V_{máx}} = 2.3\left(\frac{y}{\delta_l}\right)^{0.42}\left(1 - erf\left(0.886\frac{y}{\delta_l}\right)\right)$

where erf is the error function, V the mean velocity, V_{max} the maximum velocity in the cross section, y the depth, and δ_l the characteristic length of the hydraulic jump according to the Figure 6.

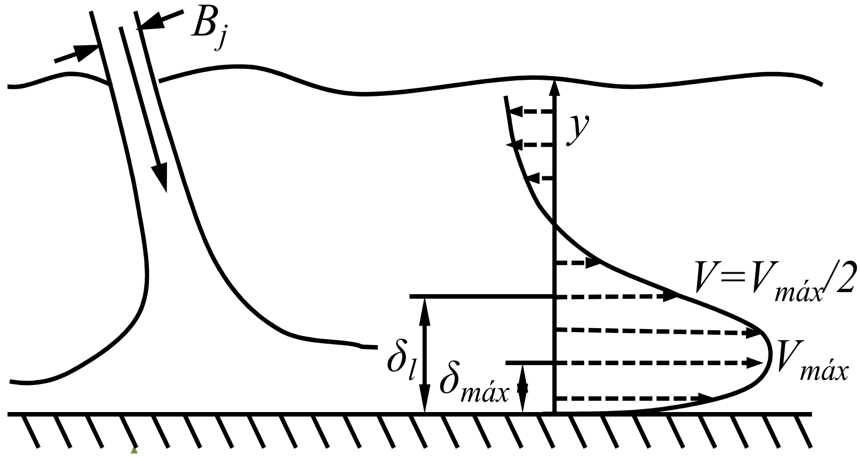

Figure 6. Schematic definition of the mean velocity profile in a submerged hydraulic jump.

Considering that V_j is the impingement velocity of the jet, B_j the jet thickness, and β the angle of the jet with the horizontal plane, Rajaratnam (1965) carried out the dimensional analysis of the Reynolds-averaged Navier–Stokes equations and obtained the basic characteristics of the flow in a stilling basin.

Figures 7 and 8 show the horizontal mean velocity data obtained in both, laboratory and numerical simulations, for a weir crest height of 2.35 m, diverse specific flows and water cushion depths together with the formulae proposed by some authors. Data have been divided by considering if the profile shows negative recirculation flow or if the entire velocity profile has direction to downstream.

The threshold between both behavior seems to be around 0.20 - 0.30 m of the stagnation point for the range of specific flows and water cushion depth analyzed. Data collapse for ratios $V_x/V_{max} \geq 0.40$. Under these value, results do not follow a single law. This is due to the jet enters into the plunge pool with an angle almost vertical, while the jet enters horizontally in the submerged hydraulic jumps downstream gates or spillways. The maximum differences among both behavior appear for the bigger water cushions (ratios $Y/B_j > 20$).

With these data, an adjustment has been proposed to define the threshold of the non-dimensional velocity profile where the recirculation region of the inverse flow tends to appear:

$$\frac{V}{V_{máx}} = 1.48\left(\frac{y}{\delta_l}\right)^{1/7}\left(1 - erf\left(0.66\frac{y}{\delta_l}\right)\right) \qquad [2]$$

where erf is the error function, δ_l the characteristic length of the velocity distribution in the hydraulic jump (depth where $V_x = V_{max}/2$).

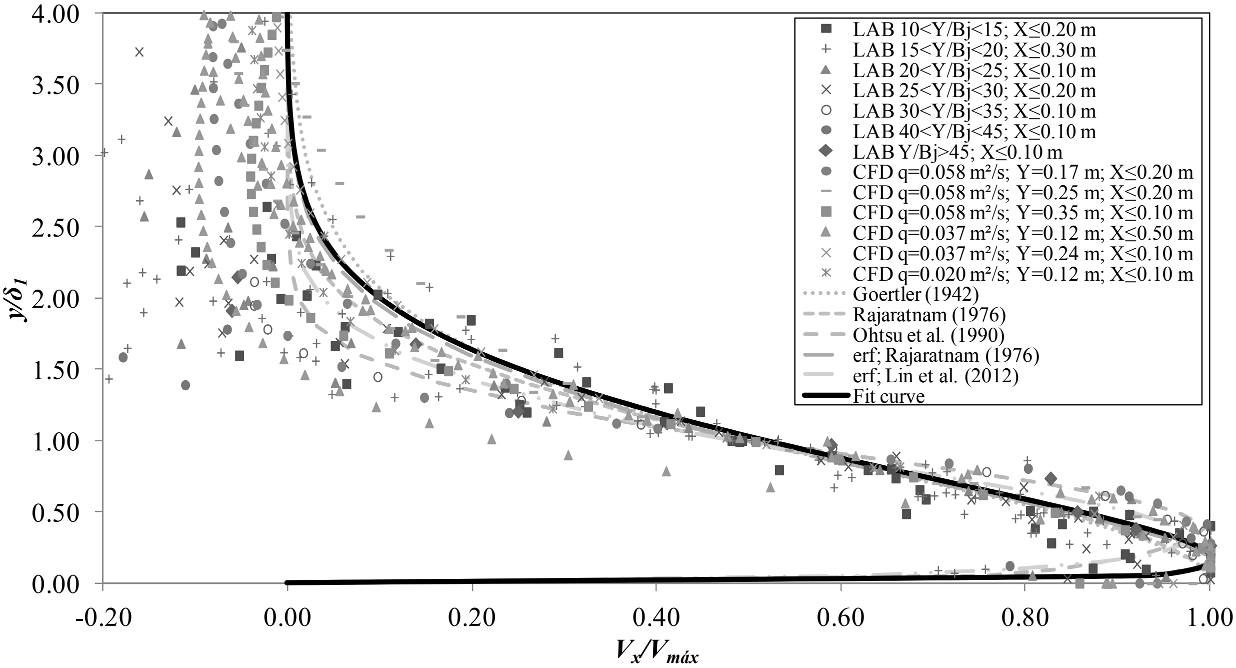

Figure 7. Distribution of the mean velocity downstream the stagnation point with laboratory data and numerical simulations. Profiles with negative flow.

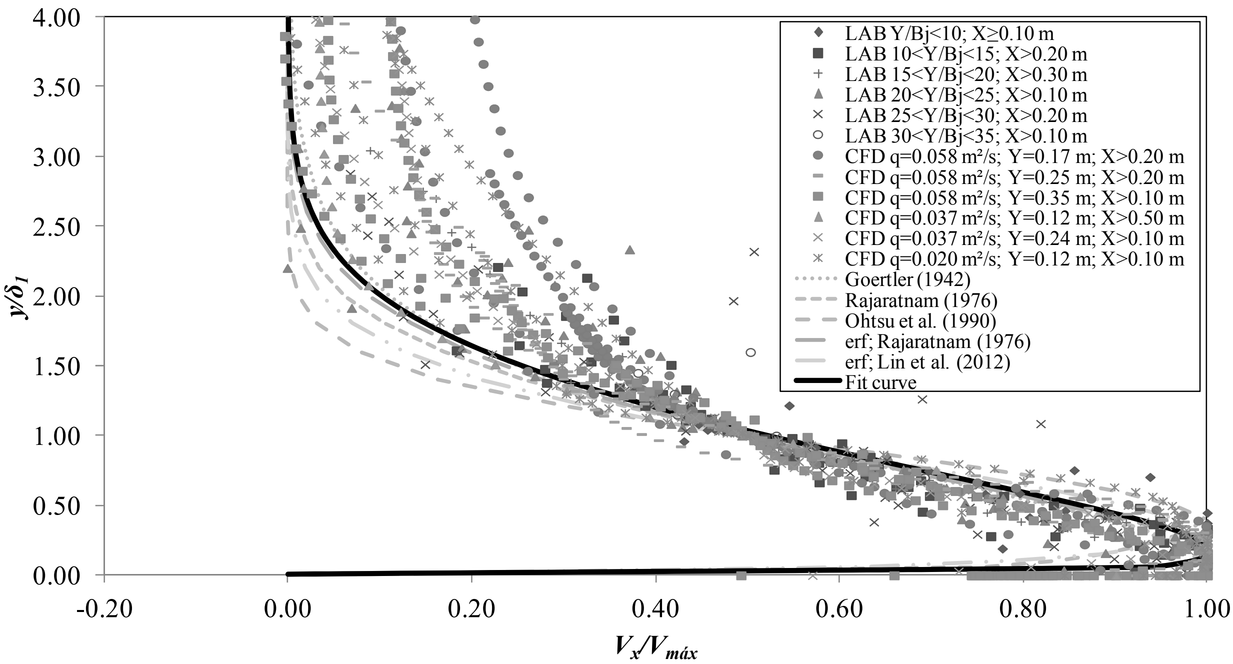

Figure 8. Distribution of the mean velocity downstream the stagnation point with laboratory data and numerical simulations. Profiles with positive flow.

5. CONCLUSIONS

The numerical results of the velocity profiles in the plunge pool downstream the stagnation point follow the laboratory data with differences smaller than 10% of the impingement velocity of the jet. However, these differences are significant in the aerated region. It was possible to adjust a velocity distribution for ratios $V_x/V_{max} \geq 0.40$. For smaller ratios it is necessary to consider different fit curves.

With the aim of improving the design of overtopping flows and their energy dissipation, it would be necessary to provide advances in the knowledge and characteristics of the hydrodynamic actions. More experimental studies, both in physical models and prototypes, are necessary in characterizing simultaneously the phenomena produced in the jets (aeration and velocity), combined with measurements of pressures, velocities and aeration rates in stilling basins.

In order to develop this work further, the researchers plan to analyze the use of inhomogeneous numerical models and laboratory measurements with high frequency (100 - 200 Hz). In future activities, comparison with diverse CFD codes (open source and commercial ones) will be considered.

ACKNOWLEDGMENTS

The researchers express their gratitude for the financial aid received from the Ministerio de Economía y Competitividad and the Fondo Europeo de Desarrollo Regional (FEDER) through the Natural Aeration of Dam Overtopping Free Jet Flows and its Diffusion on Dissipation Energy Basins project (BIA2011-28756-C03-02).

REFERENCES

Annandale, G.W. (1995). Erodibility. *Journal of Hydraulic Research,* 33(4), 471-494.
Annandale, G.W. (2006). *Scour Technology*. McGraw-Hill Professional, NY, USA.
ANSYS CFX (2011). *ANSYS CFX. Reference Guide. Release 14.0*.
Carrillo, J.M. (2014). *Metodología numérica y experimental para el diseño de los cuencos de disipación en el sobrevertido de presas de fábrica*. PhD Thesis. Universidad Politécnica de Cartagena, España.
Castillo, L.G. (2006). Areated jets and pressure fluctuation in plunge pools. *The 7th International Conference on Hydroscience and Engineering (ICHE-2006), IAHR, ASCE, Drexel University*. College of Engineering. DSpace Digital Lybrary. DU Haggerty Library. Philadelphia, USA.
Castillo, L.G. (2007). Pressure characterization of undeveloped and developed jets in shallow and deep pool. *32nd Congress of IAHR.* International Association of Hydraulic Engineering & Research, Venice, Italy, 2, 645-655.
Castillo, L.G., Carrillo, J.M., and Sordo-Ward, A. (2014a). Simulation of overflow nappe impingement jets. *Journal of Hydroinformatics*, 16 (4), 922-940.
Castillo, L.G., Carrillo, J.M., and Blázquez, A. (2014b). Plunge pool mean dynamic pressures: a temporal analysis in nappe flow case. *Journal of Hydraulic Research* (in press).
Chanson, H., and Brattberg, T. (2000). Experimental study of the air-water shear flow in a hydraulic jump. *International Journal of Multiphase Flow*, 26 (4), 583-607.
Davies, J.T. (1972). *Turbulence phenomena*. Academic Press, New York and London.
Ervine, D.A., Falvey,H.T. and Withers,W.A. (1997). Pressure fluctuations on plunge pool floors. *Journal of Hydraulic Research*, 35 (2), 257-279.
Görtler, H. (1942). Berechnung von Aufgaben der freien Turbulenz auf Grund eines neuen Näherungsansatzes. *Journal of Applied Mathematics and Mechanics / Zeitschrift für Angewandte Mathematik und Mechanik - ZAMM*, 22 (5), 244–254, (in German).
Hager, W. H. (1992). *Energy dissipators and hydraulic jump* (Vol. 8). Dordrecht, The Netherlands: Kluwer Academic Publ. Water Science and Technology Library.
Launder, B.E., and Sharma, B.I. (1972). Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. *Lett. Heat Mass Transfer,* 1 (2), 131-138.
Lin, C., Hsieh, S.-C., Lin, I.-J., Chang, K.-A., and Raikar, R.V. (2012). Flow property and self-similarity in steady hydraulic jumps. *Journal of Experiments in Fluids*, 53, 1591-1616.
Liu, P., Gao, J., and Li, Y. (1998). Experimental investigation of submerged impinging jets in a plunge pool downstream of large dams. *Science in China*, 41 (4), 357-365.
Menter, F.R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J*. 32 (8), 1598-1605.
Ohtsu, F., Yasuda, Y., and Awazu, S. (1990). Free and submerged hydraulic jumps in rectangular channels. *Report of the Research Institute of Science and Technology*. Nihon University. No 35.
Rajaratnam, N. (1965). The hydraulic jump as wall jet. *Proc. ASCE, Journal of Hydraulic Division*. 91(HY5), 107-132.
Wahl, T.L., Frisell K.H., and Cohen E.A (2008). Computing the trajectory of free jets. *Journal of Hydraulic Engineering*, 134(2), 256-260.
Wilcox, D.C. (2006). *Turbulence modeling for CFD*. 3rd edition. DCW Industries, Inc., La Canada, California.