Macián Pérez, JF.; Bayón, A.; García-Bartual, R.; López Jiménez, PA.; Vallés-Morán, FJ. (2020). Characterization of Structural Properties in High Reynolds Hydraulic Jump Based on CFD and Physical Modeling Approaches. Journal of Hydraulic Engineering. 146(12):1-13. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001820

Characterization of structural properties in a high Reynolds hydraulic jump based on CFD and physical modeling approaches

Juan Francisco Macián-Pérez[1], Arnau Bayón[2], Rafael García-Bartual[3], P. Amparo López-Jiménez[4], Francisco José Vallés-Morán[5]

[1]*Research Institute of Water and Environmental Engineering. Universitat Politècnica de València, Camí de Vera, s/n, València, 46022, Spain* (corresponding author). E-mail: juamapre@cam.upv.es

[2]*Research Institute of Water and Environmental Engineering. Universitat Politècnica de València, Camí de Vera, s/n, València, 46022, Spain*. E-mail: arbabar@iiama.upv.es

[3]*Research Institute of Water and Environmental Engineering. Universitat Politècnica de València, Camí de Vera, s/n, València, 46022, Spain*. E-mail: ragarciab@hma.upv.es

[4]*Dept. of Hydraulic Engineering and Environment. Universitat Politècnica de València, Camí de Vera, s/n, València, 46022, Spain*. E-mail: palopez@gmmf.upv.es

[5]*Research Institute of Water and Environmental Engineering. Universitat Politècnica de València, Camí de Vera, s/n, València, 46022, Spain*. E-mail: fvalmo@hma.upv.es

Abstract

A classical hydraulic jump of Fr_1=6 and Re_1= 210,000 was characterized using the Computational Fluid Dynamics (CFD) codes OpenFOAM and FLOW-3D, whose performance was assessed. The results were compared to experimental data from a physical model designed for the purpose. The most relevant hydraulic jump characteristics were investigated, including hydraulic jump efficiency, roller length, free surface profile, distributions of velocity and pressure and fluctuating variables. The model outcome was also compared to previous results from the literature. It was found that both CFD codes represent with high accuracy the hydraulic jump surface profile, roller length, efficiency and sequent depths ratio, consistently with previous research. Some significant differences were found between both CFD codes regarding velocity distributions and pressure fluctuations, although in general results are in good agreement with experimental and bibliographical observations. This makes models of these characteristics suitable for engineering applications involving design and optimization of energy dissipation devices.

Keywords: Hydraulic jump; High Reynolds number; CFD; FLOW-3D; OpenFOAM; Physical model.

INTRODUCTION

Hydraulic jumps are the sudden transition from supercritical to subcritical regime in open-channel flows. This phenomenon is characterized by its high complexity, with large turbulent fluctuations, intense air entrainment and significant energy dissipation. Despite the chaotic nature of hydraulic jumps, they are frequently classified according to their approaching Froude number (Fr_1), which establishes a relationship between flow inertial and gravity forces:

$$Fr_i = \frac{u_i}{\sqrt{g y_i}} \qquad (1)$$

where u_i is the depth-averaged velocity, g the gravity acceleration and y_i the water depth. According to Hager (1992), hydraulic jumps are considered stable for Fr_1 values ranging from 4.5 to 9.0. Higher Fr_1 values produce unstable and choppy jumps, prone to flow detachment and bubble and spray formation (Hager 1992), whereas lower Fr_1 values lead to undular jumps, characterized by lower efficiencies and formation of waves (Wang and Chanson 2015a, Liu *et al.* 2004, Chanson and Montes 1995, Chow 1959). The Reynolds number, relating flow inertial and viscous forces, also plays a crucial role when modeling flows at laboratory scale (Chanson 2009, Chanson and Gualtieri 2008). In fact, this non-dimensional number affects significantly the validity of the extrapolation of laboratory results to large size prototype hydraulic structures, due to the possible occurrence of scale effects. Reynolds numbers must be high enough to ensure model-prototype similarity (Heller 2011, Hager and Bremen 1989):

$$Re_i = \frac{u_i y_i}{\nu} \qquad (2)$$

where ν is the kinematic viscosity. The above mentioned complexity of hydraulic jumps lies on its inherent characteristics, being turbulence itself the most remarkable feature associated to such singular flow phenomenon (Jesudhas *et al.* 2018). Large-scale turbulence is found inside the roller and also at the hydraulic jump free surface, but also microscopic turbulent velocity fluctuations take place within the body of the hydraulic jump. Turbulent processes occur in a range of spatial scales and play a determinant role in the entrainment and transport of air (Jesudhas *et al.* 2016, Wang and Chanson 2015b). The effect of inflow conditions on air entrainment process was first reported by Resch and Leutheusser (1972), who addressed the turbulent structure of the hydraulic jump. Years later, Chanson and Brattbeg (2000), Chachereau and Chanson (2011) and Zhang *et al.* (2013) conducted a series of experiments that allowed characterizing the air-water structure of the hydraulic jump according to the inflow conditions. The study by Chanson and Brattberg (2000) can be considered as the first systematic experimental study of air-water flow properties in the hydraulic jump. Chachereau and Cahnson (2011) worked with several inflow Froude numbers to define the shape of the free surface profile, whereas Zhang *et al.* (2013) focused on the fluctuating nature of the impingement perimeter.

The influence of Fr_1 in the air content of hydraulic jumps was investigated by Gualtieri and Chanson (2007). These authors conducted an experimental study on the effect of Fr_1 on air entrainment. They covered Froude numbers from 5.2 to 14.3 showing that, at a fixed distance from the jump toe, the maximum void fraction increases with the increasing inflow Froude number. Witt *et al.* (2015) also studied this Froude number affection, by the means of CFD techniques. They were able to satisfactorily model velocity profiles, average void fraction and Sauter mean diameter, when compared with experimental data. Wu *et al.* (2018) also studied the inflow conditions and their affection to the air entrainment in the hydraulic jump, through an experimental campaign conducted in a hydraulic jump aeration basin. Cheng *et al.* (2017) investigated about the velocity distributions, using nonintrusive Particle Image Velocimetry (PIV) techniques, for a wide range of Froude numbers. In addition, Toso and Bowers (1988) and Mossa (1999) studied the relationship between turbulence structure and pressure and velocity fluctuations in hydraulic jumps. Toso and Bowers (1988) focused on extreme pressures, whereas Mossa (1999) experimented with several hydraulic jumps to investigate their oscillating characteristics and cyclic mechanisms. Chachereau and Chanson (2011) and Zhang *et al.* (2013) studied turbulent fluctuations and how they interfere on the hydraulic jump shape, presenting important findings on their characteristic frequencies and their length and time scale. These authors focused on the free-surface profile, while Wang and Chanson (2015a) analyzed the position of the jump toe and its fast and slow fluctuations. In both cases, the inflow conditions, and particularly the Fr_1, were paid special attention. Furthermore, Jesudhas *et al.* (2018) demonstrated the three-dimensional nature of the flow developed in the hydraulic jump. In this research, an inflow Froude number of 8.5 was considered to resolve the internal turbulent structure of the classical hydraulic jump. However, the complex interaction between the physical processes involved in hydraulic jumps, places our knowledge far from a full understanding of the phenomenon (Wang and Chanson 2015a, Wang and Chanson 2015b).

Hence, the satisfactory modeling process of hydraulic jumps remains an important challenge in many aspects. Regarding physical modeling, most of previous works focus on the measurement of external macroscopic variables, although some of them use intrusive techniques to obtain more detailed experimental data sets (Bayón and López-Jiménez 2015, Zhang *et al.* 2013). Alternatively, numerical methods, and in particular Computational Fluid Dynamics (CFD) applications, represent an interesting and useful approach to fill the gap in the modeling process of hydraulic jumps (Viti *et al.* 2019, Bayón *et al.* 2016, Bayón and López-Jiménez 2015, Castillo *et al.* 2014). A very significant research effort has been devoted to develop this kind of methods during recent years. In this respect, Langhi and Hosoda (2018) modeled a hydraulic jump with an unsteady Reynolds-Averaged Navier-Stokes (RANS) approach, obtaining satisfactory results for the free surface profile, velocity distributions and turbulence, whereas Ma *et al.* (2011), used both, a RANS and a Detached Eddy Simulation (DES) model in their simulations. These authors proved that both methods were capable to provide the void fraction profiles in the lower shear layer region. Caisley *et al.* (1999) used the software FLOW-3D to reproduce a hydraulic jump in a canoe chute. Bayón and López-Jiménez (2015), Witt *et al.* (2015) and Romagnoli *et al.* (2009) accurately modeled a hydraulic jump using the free source code OpenFOAM. These models approached a series of classical hydraulic jump variables such as the sequent depths ratio, the efficiency, the roller length and the free surface profile, providing good accuracies when compared with previous studies. Furthermore, Bayón *et al.* (2016) performed a detailed analysis of a hydraulic jump comparing the behavior of two of the most widely used codes: FLOW-3D and OpenFOAM. These authors studied the free surface profile of a classical hydraulic jump, together with the sequent depths ratio, the energy dissipation efficiency and the roller length. They also analyzed the averaged velocity field as well as the maximum velocity decay and the maximum backward velocities in the hydraulic jump roller. The results showed that both codes were able to successfully model the hydraulic jump phenomenon, despite some difficulties arose for the roller region. They also found that a quasi-periodic behavior could be observed for certain variables such as the toe or the roller end locations. It should be outlined, though, that numerical models still present some limitations to accurately reproduce certain hydraulic phenomena, as stated by Blocken and Gualtieri (2012). Consequently, the support of experimental data is crucial, and therefore physical modeling remains indispensable for a rigorous study of complex flows such as the hydraulic jump (Valero *et al.* 2019, Liu *et al.* 2018, Wang and Chanson 2015b).

Following the lines of Bayón *et al.* (2016), the aim of this research was to implement a hydraulic jump three-dimensional model in order to assess the suitability of two of the most widely spread CFD codes in hydraulic engineering applications, namely the commercial software FLOW-3D and the open source platform OpenFOAM. All results derived from both CFD platforms cited were systematically contrasted and validated using experimental data. More specifically, and for this purpose, an open-channel physical model was developed to adequately reproduce the required hydraulic jump, at the Hydraulics Laboratory of the Universitat Politècnica de València (UPV). Additionally, results from CFD modeling were compared to previous experimental works available in the literature. The case study was designed in terms of the inflow Froude (Fr_1) and Reynolds (Re_1) numbers. The first parameter (Fr_1) was set taking into account one of the most important engineering applications of hydraulic jumps, i.e., the flow energy dissipation in stilling basins (Padulano *et al.* 2017, Tajabadi *et al.* 2017). The USBR (Peterka 1964) states that hydraulic jumps with Fr_1 numbers between 4.5 and 9 provide the most efficient energy dissipation. In this case, and for the numerical models presented herein, a value of $Fr_1=6$ was adopted. Concerning the second non-dimensional number (Re_1), its choice is also very relevant due to the known limitations of physical models concerning scale effects. Although such scale effects depend on several factors, modeling the hydraulic jump with a high Reynolds number minimizes them (Heller 2011, Hager and Bremen 1989), thus providing a more reliable extrapolation of laboratory experiments. To this end, the case study analyzed herein was set up to ensure a high Reynolds number ($Re_1=210{,}000$). This makes the present research the natural continuation of Bayón *et al.* (2016), where a low-Reynolds number was analyzed employing a similar methodological basis. Hence, Bayón *et al.*, (2016) stated that the low Reynolds number ($Re_1=30{,}000$) used for their study might prevent from extrapolating their results to prototype scale structures. The present research increased the Re_1 according to the guidelines presented by Heller (2011), and also extended the experimental campaign, using improved instrumentation and measuring velocity profiles and streambed pressures. In terms of the Froude number, the same value was used ($Fr_1=6$), which falls in the previously mentioned optimal range for energy dissipation purposes (Peterka, 1964). Gathering information to characterize a hydraulic jump that can be extrapolated to prototype scale is crucial for the adaptation of existing energy dissipation structures to new scenarios, with more demanding conditions than those considered in their design.

NUMERICAL MODEL

The research presented herein assessed the performance of two CFD codes comparing their outcome when modeling the same hydraulic jump, particularly in high Reynolds Number conditions. On the one hand, version 11.0 of FLOW-3D, a commercial software package developed by FlowScience, Inc., was used. FLOW-3D works with a number of methods to model the free surface depending on the case, all of them derived from the Volume Of Fluid (VOF) as originally presented by Hirt and Nichols (1981), and has been widely used in hydraulic engineering applications since its release (Dong *et al.* 2019, Valero and Bung 2016, Caishui 2012, Sarafaz and Attari 2011, Ho and Riddette 2010). On the other hand, the case study was also modeled with OpenFOAM version 6, a CFD open platform freely available. OpenFOAM, which contains a number of C++ libraries and applications to achieve the numerical resolution of continuum mechanics problems (Weller *et al.* 1998), has also an important experience in successfully modeling hydraulic engineering problems (Teuber *et al.* 2019, Fuentes-Pérez *et al.* 2018, Bayón 2017). FLOW-3D and OpenFOAM, both based on the Finite Volume Method (McDonald 1971), were systematically compared trying to avoid bias. To this end, all model parameters were set up similarly, when possible. All discretization schemes, including those of advection equations, are second-order accurate. Regarding the run times, they are difficult to compare in this case since FLOW-3D simulations were run in a work station (where the commercial license was valid for) and OpenFOAM simulations were run in the university's HPC cluster. That makes computational times hardly comparable.

Flow equations and general settings

CFD codes base their results on the Navier-Stokes equations, which describe, in their general form, the motion of a fluid. The characteristics of the analyzed case allowed using the equations in their form for incompressible fluids. Furthermore, FLOW-3D and OpenFOAM employ the Finite Volume Method (FVM) for the spatial discretization of the conservation laws.

$$\nabla\overline{\boldsymbol{u}} = 0 \tag{3}$$

$$\frac{\partial\overline{\boldsymbol{u}}}{\partial t} + \overline{\boldsymbol{u}}\cdot\nabla\overline{\boldsymbol{u}} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\overline{\boldsymbol{u}} + \overline{\boldsymbol{f_b}} \tag{4}$$

where u is velocity, t is time, ρ is density, p is pressure and $\overline{\boldsymbol{f_b}}$ represents the body forces, namely, gravity and surface tension. Time derivatives were discretized adjusting automatically the time-step length according to the Courant number. This enhances model efficiency by reducing computational times and minimizing numerical divergence risk.

Free surface modeling

When modeling two immiscible fluids, FLOW-3D and OpenFOAM base their strategies on the Volume Of Fluid (VOF) method (Hirt and Nichols 1981), which employs an additional variable named Fraction of Fluid (F) that represents the proportion of each mesh element occupied by one fluid or another (0 is air, 1 is water). The following expression is used to compute the evolution of F throughout the domain:

$$\frac{\partial F}{\partial t} + \nabla \cdot (\overline{\boldsymbol{u}}F) = 0 \qquad (5)$$

The VOF method covers the transport of other properties (e.g. ξ) by means of weighted averages, according to the value of F in each mesh element:

$$\xi = \xi_{water}F + \xi_{air}(1 - F) \qquad (6)$$

The VOF approach, as described above, leaves an unresolved question: defining a neat fluid interface in regions where F values are between 0 and 1. OpenFOAM works around this problem by introducing a fictional velocity term to Eq. 5 ($\nabla \cdot (\bar{u}_c \, F[1 - F])$). This summand adds a fictional velocity in the direction of the largest gradient of F, which tends to "compress" the air-water interface, as depicted in Bayón *et al.* (2018). In regards with FLOW-3D, under a two-fluid approach, this code simultaneously solves a set of conservation equations for each phase separately. For fluids greatly differing in their densities and separated by a thin interface, such as the ones presented in this research, a free-slip velocity condition at the interface is recommended. By adding this condition, the momentum coupling could be improved. Furthermore, a mechanism was added to help close up partial voids and add interface sharpening to preserve the free surface and improve its tracking. This so-called F-packing mechanism works by creating small negative divergences in internal fluid cells.

Air entrainment

Aeration is a crucial phenomenon in highly turbulent air-water flows. Eddies and free surface fluctuations cause air entrapment, thus forming bubbles in the hydraulic jump body (Xiang *et al.* 2014). The presence of air affects the momentum transfer as it modifies the flow macroscopic density, adds compressibility, increases its depth and induces volume bulking (Chanson 2013, Favley 1980). Consequently, an accurate approximation to the air entrainment phenomena becomes an important issue when modeling a hydraulic jump. Along the same lines, it is worth considering that water droplets and air bubbles may show a characteristic length scale below the mesh size, making its tracking considerably difficult (Bayón *et al*. 2016, Valero and Bung 2015; Lobosco *et al*. 2011).

Air entrainment could be modeled by establishing a balance between stabilizing forces (gravity and surface tension) and destabilizing forces (turbulence). This allows a continuous estimation of the rate at which air enters the flow. However, multiple input parameters are needed for such detailed modeling process. Besides, calibration and validation of these parameters is also necessary. Instead of that, for this particular study in which aeration and void fraction distributions were not analyzed, an entirely Eulerian method with two fluids, similar to the one referred by Bayón *et al*. (2016), was used as a modeling approach. Hence, both fluids were allowed to mix in the same cell, but locating the free surface where F=0.5. Nevertheless, no additional equations were used for droplet and bubble dynamics.

Turbulence modeling

Modeling turbulence is one of the key aspects of CFD applications. At high Reynolds numbers, the natural instabilities that occur within the flow lead to swirling structures of different scales. Ideally, velocity and pressure fluctuations derived from turbulence would be resolved to their lower scale through the so-called Direct Numerical Simulation (DNS) approach. However, this is not practical in most applied cases, due to computer memory and processing time limitations. Therefore, the majority of CFD applications incorporate a turbulence model to describe and quantify the effects of turbulence on the mean flow characteristics. The Large Eddy Simulation (LES) method is based on the direct resolution of the largest turbulent structures and the modeling of those below a certain scale. Generally, this is an accurate approximation to reality, but still unaffordable for most engineering applications (Bayón *et al*. 2016, Spalart 2000). Finally, the Reynolds Averaging of the Navier-Stokes Equations (RANS) is probably the most popular approach for engineering problems. RANS models find closure to the turbulence problem by averaging the so-called Reynolds stresses and adding supplementary variables related to the turbulent viscosity and their respective transport equations. There are different turbulence models according to the number of additional transport equations used to solve the closure problem. Two-equation models are the most frequent option, as they are able to provide a full description of turbulence in terms of time and length scales and hence reproduce a wide range of flows (Pope 2000).

For the CFD models set up in the present study, a two equation RNG k-ε turbulence model (Yakhot *et al*, 1992) was used. The RNG k-ε approach applies statistical methods to the derivation of the averaged equations for two turbulence quantities: turbulent kinetic energy (k) and its dissipation rate (ε). One of the advantages of this model is that usually provides better results when modeling swirling flows compared to the standard k-ε (Bombardelli *et al.* 2011, Kim and Baik 2004, Pope 2000, Bradshaw 1996, Speziale and Thangam 1992). The transport of k and ε was modeled by the following two equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\rho\varepsilon \tag{7}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}P_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{8}$$

where x_i is the coordinate in the i axis, μ is dynamic viscosity, μ_t is turbulent dynamic viscosity and P_k is the production of TKE. Finally, the terms σ_k, σ_ε, $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are parameters whose values are given in Yakhot *et al.* (1992).

Geometry and meshing

The spatial domain subject of the present study consisted of a horizontal rectangular channel where a classical hydraulic jump takes place. The simplicity of the geometry favored the use of a structured mesh. According to Biswas and Strawn (1998) and Hirsch (2007), models using these meshes generally provide a better accuracy than those using unstructured meshes and their generation algorithms are faster and show a lower complexity degree. In addition, structured meshes have associated a more regular access to memory and consequently the latency during simulations is lower (Keyes *et al.* 2000). Finally, numerical diffusivity in free surfaces tends to be reduced when modeling multiphase flows with topologically orthogonal meshes (Bayón and López-Jiménez 2015).

Unstructured meshes show multiple advantages such as their capability to refine selectively regions where important gradients of the flow variables are expected (Kim and Boysan 1999). Besides, their arbitrary topology can not only adapt better to complex geometries, but also produce fewer closure issues (Biswas and Strawn 1998). However, given the abovementioned simplicity of the geometry of the case under study, none of the advantages of unstructured meshes constituted a significant improvement. Therefore, a structured rectangular hexahedral mesh was used. In the meshing process, areas where there was no flow were cropped and a cell refinement in those other areas where higher flow gradients were expected was carried out, looking for efficiency in the simulation process without affecting the results (Figure 1). Thus, two different cell sizes were used, being the cell size relation between them 1:2, for the three space coordinates (i.e. 1:2 relation for Δx, Δy and Δz). The mesh elements size was determined through a mesh convergence analysis, which is developed in forthcoming sections.

Boundary conditions

Boundary conditions were set up so that the hydraulic jump took place within the modeled channel stretch. Consequently, a supercritical flow inlet and a subcritical flow outlet were imposed. The appropriate value of Fr_1 was ensured with a constant flow depth at the inlet (y_1) and the corresponding velocity value computed according to Eq. 1. A subcritical boundary condition was imposed to the outlet, varying its flow depth (y_2) iteratively in order to place the hydraulic jump on the desired position. In respect with the inlet variables for the RANS model, namely k and ε, they were set to small values arbitrarily so that they developed as the simulation progressed, since their initial value was unknown. Furthermore, the wall roughness was neglected in consistence with the small roughness of the materials of the experimental device walls and streambed (glass and PVC, respectively). A high Reynolds number wall function was imposed to the solid contours, thus allowing a significant saving in computational costs. In order for the function to operate properly, it must be ensured that the y^+ coordinate of all elements in contact with solid boundaries remains below $y^+<300$. The computation of y^+ is based on the non-dimensionalization of velocity profiles according to the shear velocity (u_τ) proposed by von Kármán's (1930) Law of the Wall:

$$y^+ = y\frac{u_\tau}{\nu} \tag{9}$$

$$u^+ = \frac{u}{u_\tau} \tag{10}$$

EXPERIMENTAL SETUP

The results obtained with the numerical models were compared, not only to previous studies, but also with the authors' own experimental results. In order to carry out this comparison, an open channel, installed at the Hydraulics Laboratory of the Universitat Politècnica de València (UPV), was used. This rectangular-section channel is built with a PVC streambed and glass walls, and its dimensions are 10.00 m long, 0.30 m wide and 1.00 m high. The inlet to the system is a pressure flow, with a transition between pressure and free surface flow right before the entrance of the channel. The channel pump allows discharges up to 140 l/s, enough to reproduce a wide range of Froude numbers. The flow rate was controlled by an electromagnetic flow meter by SIEMENS © (SITRANS MAG5100 W), able to measure flow rates between 1 m^3/h and 2500 m^3/h, with an uncertainty < 0.1%. The channel is also equipped with both, upstream and downstream sluice gates, which can be maneuvered to control the supercritical and subcritical flow depths. This experimental device was equipped with pressure and temperature sensors. The longitudinal axis of the channel streambed presents holes each 50 cm to host pressure sensors, which remain blocked when they are not in use to ensure the water tightness of the channel. Hence, pressure transmitters could be located in multiple exchangeable positions all along the channel. These transmitters, with the corresponding software, allowed to record quick and precise pressure and temperature data thanks to their piezoresistive transducer and microprocessor with 16 bit A/D converter.

Velocities were measured using both an Acoustic Doppler Velocimeter (ADV) (Vectrino, Nortek) and a Pitot tube (PASCO General flow sensor with Pitot tube). The ADV allows measuring the three components of the velocity vector in a point using the Doppler Effect. This device, which offers data collection rates up to 200 Hz, is designed to cover a range of velocities from 3 cm/s to 4 m/s, in conditions where the signal is not affected by flow elements such as bubbles. In the present research, it was used to measure flow velocity distributions downstream of the hydraulic jump roller. A back-flush Pitot tube for velocities up to 10 m/s was employed to measure velocities larger than 4 m/s both upstream and within the hydraulic jump roller. The roller length was measured using the stagnation point criterion: vertical profiles of average streamwise velocity were measured in several sections along the roller. The point where velocity tends to zero (stagnation point) was identified in all of them. Finally, the intersection between the line joining all these points and the average free surface indicated the end of the roller position (Hager *et al.* 1990).

In respect with the definition of the free surface profile in the physical model, different techniques were employed depending on the area where measures were taken. The use of a varied methodology to measure the hydraulic jump profile provides a deeper contrast, which in turn is relevant in the analysis due to the high level of turbulence in the phenomenon, leading to a randomly variable free surface (Castro-Orgaz and Hager 2009). The instant and average free-surface profiles throughout the hydraulic jump were obtained from the experimental channel using digital image processing (DIP). An edge detection method based on a light intensity threshold allowed to identify air-water interfaces in videos of the hydraulic jump profile recorded at 50 Hz with a resolution of 1280x720 px. The quality of the results was enhanced by applying perspective effect correction and filtering algorithms to remove the bias caused by droplets, reflections and others. Free-surface position was also recorded at several points along the hydraulic jump using HC-SR04 ultrasound distance meters connected to a Raspberry Pi 3 B+. In addition, point gauge measurements using limnimeters were conducted throughout the channel in order to contrast the DIP and ultrasound sensor results. Hence, the experimental campaign comprised not only the hydraulic jump roller, but also the flow upstream and downstream, with the purpose of achieving a characterization of the hydraulic jump as accurate as possible.

It is important to remark that the flow under study is extremely complex. Thus, obtaining reliable measures of certain variables remains a challenging goal, given the available measuring devices and experimental limitations. According to Valero *et al.* (2019), even a perfectly sampled data series could still present uncertainties related to the limitations on the measuring time or the data acquisition rate. As a result of this, there was an unavoidable degree of uncertainty associated with the parameters studied in the experimental campaign. However, its design was made seeking for a reduction of this uncertainty, choosing appropriate measuring times and locations. In addition, the corresponding preliminary analyses were conducted, in order to discard anomalous data or those other values lacking of physical sense.

CASE STUDY

The comparison carried out between the CFD codes was based on a particular case study of a three-dimensional classical hydraulic jump tested in the laboratory open flow channel above referred. Discharge in the channel was set to Q=0.063 m^3/s (discharge per unit width: q=0.21 m^2/s) and the supercritical flow depth was y_1=0.05 m (u_1=4.2 m/s). These values led to an inflow Froude number of Fr_1=6, a Reynolds number of Re_1= 210,000 and a Weber number of We_1=12,058. The Weber number, proportional to the ratio of the inertial to surface tension forces is calculated as (Chanson 2006):

$$We_i = \frac{\rho u_i^2 y_i}{\sigma} \tag{11}$$

where σ is the surface tension coefficient. Regarding the characteristics of the fluids, for water, the density and kinematic viscosity were respectively ρ_w=998 kg/m^3 and ν_w=10^{-6} m^2/s, whereas for air ρ_a=1.184 kg/m^3 and ν_a=1.781^{-5} m^2/s were used. The surface tension coefficient was σ=0.073 N/m.

Mesh convergence analysis

A mesh convergence analysis was carried out on both CFD codes to determine the appropriate cell size for the case study. That ensures the independence of the numerical model results from the size of cell implemented or, at least, quantifies the result numerical uncertainty. The analysis was conducted following the ASME's criterion (Celik *et al.* 2008), using four different meshes and twenty one indicator variables (streamwise flow velocity at different points within the roller). The cell sizes tested in the different meshes were 1.67, 1.25, 1.00 and 0.71 cm, accomplishing the recommended approximate minimum ratio between coarsest and finest meshes of 1.3 (Celik *et al.* 2008). The data of FLOW-3D showed limited influence on results compared to OpenFOAM, as previously observed by Bayón *et al.* (2016). This smaller sensitivity made the mesh convergence analysis perform worse, so the best results were achieved by mesh size Δx=0.71 cm, with a mesh apparent order of p=1.96, near the model formal order, and a grid convergence index of GCI=63.5%. The latter value indicates a large numerical uncertainty, in coherence with the reduced sensitivity to mesh refinement. OpenFOAM, on the contrary, showed a clearer convergence process, which normally makes more refined meshes necessary to achieve reliable results. Coherently, the best results were yielded by mesh size Δx=0.50 cm, with an apparent order of p=2.2, slightly above the model formal order, and a convergence index of GCI=11.6%. This significantly smaller GCI value indicates a clearer path to convergence and smaller levels of numerical uncertainty. However, the indicators for a mesh size Δx=0.71 cm were also satisfactory for OpenFOAM. Hence, prioritizing a similar model set up, this was the cell size used in both codes.

Stability of the solution

Given the chaotic nature of the flow studied, the variables describing the phenomenon were averaged in time windows long enough to ensure stationarity. To this end, it is important to run simulations until the quasi-stationary state is reached, thus allowing a proper statistical result averaging. In this respect, simulations were performed to attain the desired position of the hydraulic jump. After that, a 10-second simulation in which the variation on the fluid fraction in the domain is under 3.5% was used for averaging and the subsequent analysis.

RESULTS AND DISCUSSION

The observation of the simulations performed by both CFD codes showed that they were able to reproduce the studied phenomenon in a physically-consistent way. The hydraulic jump occurred in the desired position and the macroscopic qualitative features, such as the subcritical and supercritical flow, the high vorticity in the roller area, the gradual air detrainment downstream of the hydraulic jump toe, etc. were in good agreement with those expected for a classical hydraulic jump (Viti *et al.* 2019, Hager 1992). A thorough analysis of some of the characteristic variables of hydraulic jumps is conducted hereunder.

Hydraulic jump characterization

The sequent depths ratio obtained with the two CFD models employed was 7.46 for FLOW-3D and 7.50 for OpenFOAM. The accuracies obtained when comparing these values to the experimental results yielded with different techniques (DIP, limnimeters and ultrasound sensor) were 94.2, 96.4 and 96.4% for FLOW-3D and 94.7, 96.9 and 96.9% for OpenFOAM. Furthermore, using the expression proposed by Hager and Bremen (1989), which is based on the Bélanger equation (Bélanger 1841), the accuracies were 96.5 and 97.0% respectively for FLOW-3D and OpenFOAM. It is important to highlight that for the physical model, the different techniques employed gathered high accuracies in the determination of the sequent depths ratio compared to Hager and Bremen (1989). Overall, the results were quite similar for both CFD codes, in good agreement with the experimental results and Hager and Bremen (1989); although the accuracies were slightly lower for FLOW-3D. Regarding the hydraulic jump efficiency, as described by Hager (1992): $\eta = \left(1 - \sqrt{2}/Fr_1\right)^2$, FLOW-3D and OpenFOAM yielded an accuracy of 99.0% and 99.3% respectively, whereas when comparing with the hydraulic jump physical model, the accuracy was 97.7% for FLOW-3D and 98.0% for OpenFOAM, in relation with the data gathered with limnimeters and ultrasound sensor; and 96.1% and 96.4% respectively for the DIP. High levels of accuracy for the hydraulic jump efficiency were expectable since this variable, which gives a measure of the amount of energy dissipated in the hydraulic jump, is strongly correlated to the sequent depths ratio.

Free surface profile

The free surface profile is an important aspect of hydraulic jumps that has been widely studied in the past (Wang & Chanson 2015a, Castro-Orgaz & Hager 2009, Bakhmeteff & Matzke 1936). Figure 2 displays the averaged dimensionless free surface profile for both CFD codes, along with results from the experimental campaign and other authors' data (Wang & Chanson 2015a, Bakhmeteff & Matzke, 1936). To obtain the dimensionless profile the following expressions were used:

$$X = \frac{x - x_0}{L_r} \tag{12}$$

$$Y = \frac{y - y_1}{y_2 - y_1} \quad (13)$$

where x_0 is the hydraulic jump toe position, y_1 and y_2 the supercritical and subcritical flow depths and L_r the hydraulic jump roller length. Both numerical models were able to reproduce the free surface profile of the hydraulic jump as their profiles mostly fall between the ones proposed by Wang and Chanson (2015a) and Bakhmeteff and Matzke (1936). It can be remarked that there was a slight overestimation of Y in comparison with the Bakhmeteff and Matzke (1936) profile for $X > 1$, but this is in good agreement with the results reported in Hager (1992) and Bayón *et al.* (2016). Furthermore, the experimental profile obtained with DIP was in good agreement with the rest of the results but in general, tended to overestimate the flow depths. This overestimation can be consequence of the bias caused by bubbles influencing the digital image treatment in a phenomenon where free surface turbulence and air entrapment play a significant role, and droplets and bubbles are continuously expelled. Moreover, numerical models provided the free surface profile along the longitudinal axis of the hydraulic jump, whereas DIP techniques must take images from the side of the experimental channel and consequently, the free surface instant rotation around the X axis can affect the results. These factors would explain the resulting overestimation of flow depths. In order to minimize these differences, further research is needed. Regarding the point measurements obtained with the ultrasound sensor, the results improved as they moved downstream from the hydraulic jump toe, until they achieved a high accuracy level for the subcritical regime. Consequently, it seems that high velocities, bubble and droplet ejection, and intense free surface turbulence affected the sensor reliability. The coefficient of determination R^2 (Bennet *et al.* 2013) was calculated to assess the accuracy of the numerical models. Hence, FLOW-3D achieved a value of R^2=0.991 compared with Bakhmeteff and Matzke (1936), R^2=0.956 compared with Wang and Chanson (2015a) and R^2=0.943 in relation with the experimental results (DIP), whereas for OpenFOAM the coefficients of determination were R^2=0.996, R^2=0.996 and R^2=0.961 respectively. Taking into account that R^2=1 indicates a perfect agreement, the models here presented were able to reproduce accurately enough the free surface profile of the hydraulic jump.

Velocity profiles

Velocity profiles in different positions along the hydraulic jump longitudinal axis were obtained, averaged and analyzed. Figure 3 a) shows that the maximum velocity decay from the hydraulic jump toe followed a similar trend in OpenFOAM and FLOW-3D, which is in good agreement with the expression proposed by Hager (1992) and the experimental data. In the latter case, the trend showed a higher degree of variability, likely due to the possible bias suffered by the Pitot tube in the swirling region of the hydraulic jump. It is important to remark that to experimentally obtain the maximum velocity at a certain location, vertical profiles were measured along the hydraulic jump longitudinal axis. From these measures, the maximum streamwise velocity in each profile was extracted. The coefficient of determination (R^2) was calculated for the CFD models in relation with Hager (1992), resulting in the highest accuracy for FLOW-3D (0.999), followed by OpenFOAM with a value of 0.992. For the maximum backward velocity, the differences between the models and the values reported by Hager (1992) increased (Figure 3 b)), probably as a result of the complex flow taking place in the area with recirculation. Hence, R^2 was 0.928 for OpenFOAM and 0.618 for FLOW-3D compared to Hager's results, whereas experimental data seemed to follow a trend closer to FLOW-3D results. Bayón *et al.* (2016) also observed similar discrepancies between FLOW-3D, OpenFOAM and Hager (1992). The maximum and maximum backward velocity dimensionless values were obtained as stated by Hager (1992).

In regards with the vertical velocity profiles in the roller region, the information obtained from the numerical and the physical models was compared with the analytical expression proposed by McCorquodale & Khalifa (1983), which represents the mean velocity distribution using two different functions that distinguish between inner and outer layer:

$$u = u_{max}\left(\frac{z}{\delta}\right)^{1/7}; 0 \leq z \leq \delta \tag{14}$$

$$u = u_{\infty} + u_t e^{2.772(z-\delta/y-\delta)^2}; \delta < z < y \tag{15}$$

where u is the horizontal velocity, u_{max} is the maximum horizontal velocity, which takes place at a height $z = \delta$, u_{∞} is the horizontal component of the freestream velocity and $u_t = u_{max} - u_{\infty}$. The adjustment of this expression to the values obtained in the models allowed presenting dimensionless results, following the procedure found in Hager (1992) for the diffusion portion of the velocity profile, as shown in Figure 4. This figure also includes the theoretical dimensionless expression for velocity profiles in the hydraulic jump roller region proposed by Hager (1992):

$$U = [cos(100Z)]^2 \tag{16}$$

A general observation of Figure 4 shows a good agreement of the CFD models with the theoretical results proposed by Hager (1992), with a slightly steeper velocity decay in the velocity values for OpenFOAM compared to literature results. In respect with FLOW-3D, the analyzed profiles increased their differences with the theoretical expression as they approached the end of the hydraulic jump roller, where the velocity distribution proposed by Hager (1992) may not be strictly followed. In spite of this, the profiles were almost coincident with the bibliographic results, at least until $Z \sim 0.6$. For larger Z values, although the results still showed a satisfactory agreement with the expression by Hager, the differences slightly increased. This result implies that the ability showed by FLOW-3D to accurately reproduce the velocity field within the jump roller, diminished as the profiles approached the free surface. This is precisely the zone where backwards velocities gain importance, a fact that basically explains the differences found in Figure 3 b) between FLOW-3D, OpenFOAM and the bibliographic results. However, both numerical models accurately reproduced vertical velocity distributions along the hydraulic jump roller compared to Hager (1992) with a R^2 coefficient of 0.984 for FLOW-3D and 0.978 for OpenFOAM.

Regarding the experimental values, higher differences with the expression by Hager (1992) were observed. Firstly, as explained for FLOW-3D, the profile with the highest X value did not strictly follow the analytical expression, probably due to its proximity to the roller end section. For the rest of the profiles, despite the general good agreement observed for low Z values, the differences increased for $Z > 0.5$. The most probable explanation to such differences concerns the Pitot tube measurements reliability. It can be considered reliable in the bottom area of the jump, where air concentrations are relatively low, but its accuracy decreases significantly inside the highly aerated region, close to the free surface (Wang 2014). This explanation is also suitable for the results observed in Figure 3 b), considering that the highest presence of bubbles within the hydraulic jump is generally associated to those areas where the maximum backwards velocities take place.

Velocity profiles in the supercritical and the subcritical flow regime were also analyzed both for the numerical and the physical models. A comparison of these results with the analytical expression (Eq. 17) proposed by Kirkgoz and Ardiclioglu (1997) is shown in Figure 5.

$$\frac{u}{u^*} = 2.5 \cdot \ln\left(\frac{zu^*}{\nu}\right) + 5.5 \qquad (17)$$

where $u*$ is the shear velocity, as estimated by the same authors. Values from the physical and the numerical FLOW-3D and OpenFOAM models were around the expected results according to the expression from Kirkgoz and Ardiclioglu (1997). On the one hand, for the supercritical regime, a quicker increase in the velocity values could be observed for the models when compared with bibliographical results. On the other hand, for the subcritical regime, the trends defined by the numerical models seemed to differ from the rest of the results. This was probably due to the proximity of the analyzed sections to the hydraulic jump roller (lower values of *X*). The roller affects these velocity profiles, so that they were closer to bibliographical expressions referred to velocity profiles in this region, such as the above mentioned Hager (1992) velocity profile (Eq. 14). Therefore, Hager (1992) was the comparison source displayed for these profiles in Table 1. In the numerical models, profiles with higher *X* values were affected by the downstream boundary condition. Consequently, they were not analyzed.

Pressure

An analysis of the pressures in the streambed was conducted for the numerical and experimental models. The averaged relative pressures along the hydraulic jump longitudinal axis were compared to observations from Toso and Bowers (1988) in a classical hydraulic jump with Fr_1=5.67. Figure 6 a) shows a good agreement between the numerical models and the observations from Toso and Bowers (1988), leading to a value of 0.995 of the coefficient of determination for FLOW-3D and 0.958 for OpenFOAM. The experimental results showed a high variability and it was difficult to find trends or similarities with the numerical models or the bibliographical data. It is important to highlight that pressure transmitters are highly sensitive to solid particles carried in the flow, which could have affected the results. Apart from these values, pressure fluctuations, which are closely related to the turbulent nature of the jump, were analyzed. To this end, the procedure proposed by Abdul Khader and Elango (1974) was followed, decomposing pressure instant values into: $p = \bar{p} + p'$, where $\bar{p}$ is the average value and p' is the fluctuating component. Hence, pressure fluctuations could be obtained as P/P_m, where:

$$P = \frac{\sqrt{p'^2}}{\rho u_1^2 /_2} \tag{18}$$

$$P_m = a(1 + aFr_1) \tag{19}$$

with a=0.061 for the domain 4.7<Fr_1<6.6. Figure 6 b) displays a distribution of the pressure fluctuations within the hydraulic jump. Results from both CFD models showed a similar trend to the observations made by Abdul Khader and Elango (1974) but with a lower value of the maximum fluctuations, followed by a slight overestimation of these fluctuations for $X > 0.5$. In terms of the position where the peak pressure fluctuations take place, the results from FLOW-3D and the experimental model were close to the observations made by Toso and Bowers (1988), which indicated a position around X=0.4 for this peak, whereas OpenFOAM was in the line of other bibliographical results which determined X=0.3-0.35 as the position where the maximum pressure fluctuations occur (Spoljaric 1984, Abdul Khader and Elango 1974).

Roller length

As defined by Hager *et al.* (1990), the hydraulic jump roller marks the boundary between backward and forward flow, starting at the toe of the jump and ending at the surface stagnation point. In order to obtain the roller length, the stagnation point criterion, as described in previous sections, was applied for both the physical and the numerical models. Hager *et al.* (1990) carried out an extensive literature review on roller lengths, measured for different hydraulic jumps, and proposed the following expression to obtain it:

$$L_r = y_1\left[-12 + 100\,tanh\left({}^{Fr_1}/_{12.5}\right)\right] \qquad (20)$$

Furthermore, Wang and Chanson (2015a) proposed an expression based on their observations for hydraulic jumps with a value of the inflow Froude number between 1.5 and 8.5:

$$L_r = y_1[6(Fr_1 - 1)] \qquad (21)$$

The hydraulic jump roller length for the FLOW-3D, OpenFOAM and physical models was 1.40 m, 1.59 m and 1.57 m respectively, whereas a value of 1.63 m and 1.50 m was obtained using Eqs. (20) and (21). OpenFOAM appears to be more precise in the estimation of this parameter as it yielded accuracies of 98.7%, 97.5% and 94.0% in comparison with the physical model and results from Hager *et al.* (1990) and Wang and Chanson (2015a) respectively, whereas for FLOW-3D these accuracies were 89.2%, 85.9% and 93.3%. Despite the differences, both models achieved an acceptable accuracy for this parameter, which turns out to be crucial when modeling a hydraulic jump as it limits the region where the biggest pressure and velocity fluctuations and the largest energy dissipation occur. Correct roller estimation is hence of utmost importance when designing energy dissipation structures where hydraulic jumps take place. Finally, a table summarizing the performance of the CFD models developed in this research in comparison with experimental data and bibliographical results is displayed (Table 1).

CONCLUSIONS

The hydraulic jump is known to be one of the most complex phenomena in fluid mechanics. The research here presented provides an insight on several structural properties of a hydraulic jump of Fr_1=6 and Re_1=210,000. The definition of the phenomenon with these values of Fr_1 and Re_1 minimizes scale effects (Heller 2011) and thus, provides a reliable extrapolation of the experiments performed, which are representative of the preferable hydraulic jumps taking place in stilling basins (Peterka 1964). The capability of the CFD codes OpenFOAM and FLOW-3D to model this hydraulic jump was assessed and contrasted with experimental data from a specifically designed physical model and bibliographical results from an extensive literature review. The most relevant conclusions obtained are stated below.

The hydraulic jump efficiency and the sequent depths ratio were accurately reproduced by the CFD codes when compared to both, laboratory measurements and previous literature results. In terms of the free surface profile, the presence of bubbles and droplet ejection, combined with the intense free surface turbulence, introduced a degree of bias in the fluctuating profile. These affected particularly physical measurements (DIP and ultrasound sensors), which slightly overestimated depths in the profile. Meanwhile, both numerical codes yielded similar results, in good agreement with other research published on this topic (Wang and Chanson 2015a, Bakhmeteff and Matzke 1936).

Concerning velocity distributions, a comprehensive analysis was carried out, including maximum velocity decay from the hydraulic jump toe, maximum backward velocity, and velocity profiles characterization in three different areas, namely, inner roller region, supercritical and subcritical flow regimes. Maximum velocity decay was successfully reproduced by both CFD codes. However, for the maximum backward velocity, significant differences were found. FLOW-3D results were closer to the experimental ones, whereas OpenFOAM was in better agreement with the results reported by Hager (1992). Regarding velocity profiles, the supercritical non-dimensional velocity distribution pattern was well reproduced by both numerical codes, when compared to experimental results and also to the analytical profile proposed by Kirkgoz and Ardiclioglu (1997), with the highest R^2 values reached by FLOW-3D in this case. On the other hand, there were significant discrepancies in the velocity profile patterns for the subcritical flow.

For this regime, OpenFOAM distributions showed an important curvature with a distinguished relative maximum and FLOW-3D generated a profile closer to a uniform distribution, whereas experimental results provided almost linear profiles with maximum velocities close to the surface. These differences could be explained by the different *X*-sections analyzed, which were chosen taking into account restrictions of both, the numerical and the experimental approach. In respect with the velocity distributions along the hydraulic jump roller, numerical outputs reached a high level of precision, when compared to Hager (1992). It should be pointed out that for both CFD codes, non-dimensional velocity profiles were almost coincident, no matter the chosen section inside the roller. On the contrary, experimental results variations were more relevant, depending on the *X*-section considered.

Pressure distributions in the channel bed were very accurately reproduced by the numerical models, with results almost coincident with those reported by Toso and Bowers (1988). Regarding pressure fluctuations, which were quantified according to Abdul Khader and Elango (1974), OpenFOAM results were close to previous reported research on the topic, whereas FLOW-3D outputs showed a significant overestimation for sections $X > 0.5$. For the roller length estimation, which is a crucial matter in several hydraulic jump applications, OpenFOAM outcomes were in good agreement with experimental results and previous research on the topic (Wang and Chanson 2015a, Hager *et al.* 1990) while FLOW-3D, even providing acceptable results, slightly underestimated this variable.

The comparisons made showed that the numerical approach using FLOW-3D and OpenFOAM was able to adequately reproduce the main structural properties of the hydraulic jump, although they failed to represent some internal details with total accuracy. It is important to highlight that knowledge and understanding of the hydraulic jump at its different scales remains limited and therefore, potential advances achieved with CFD techniques constitute a promising research line, which is worth developing. The research here presented concludes that the considered CFD codes can successfully complement experimental modeling and literature to analyze hydraulic jump characteristics on prototype structures. Consequently, the yielded results may help to improve the performance of energy dissipation structures in dams. The adaptation of these structures to higher discharges than those considered in the design phase is in the spotlight, due to climate change effects and increasing society demands in terms of security. Hence, contributions to hydraulic jump modeling, as the ones presented in this research, are crucial to face the challenge of energy dissipation structures adaptation.

DATA AVAILABILITY STATEMENT

All data, models and code generated or used during the study appear in the submitted article.

ACKNOWLEDGEMENTS

The research presented herein was possible thanks to the 'Generalitat Valenciana predoctoral grants (Ref. [2015/7521])', in collaboration with the European Social Funds and to the research project: 'La aireación del flujo y su implementación en prototipo para la mejora de la disipación de energía de la lámina vertiente por resalto hidráulico en distintos tipos de presas' (BIA2017-85412-C2-1-R), funded by the Spanish Ministry of Economy.

REFERENCES

Bakhmeteff, B. A., and Matzke, A. E. 1936. "The hydraulic jump in terms of dynamic similarity." *Trans. ASGE'101*, 101(1), 630-647.

Bayon, A., Macián-Pérez, J. F., Vallés-Morán, F. J., López-Jiménez, P. A. 2019. "Effect of RANS turbulence model in hydraulic jump CFD simulations." In *E-proceedings of the 38th IAHR World Congress*. Panama City, Panama.

Bayon, A., Toro, J. P., Bombardelli, F. A., Matos, J., and López-Jiménez, P. A. 2018. "Influence of VOF technique, turbulence model and discretization scheme on the numerical simulation of the non-aerated, skimming flow in stepped spillways." *J. Hydro-environ. Res.*, 19, 137-149.

Bayon, A. 2017. "Numerical analysis of air-water flows in hydraulic structures using computational fluid dynamics (CFD)." PhD Thesis. Universitat Politècnica de València, Spain.

Bayon, A., Valero, D., García-Bartual, R., Vallés-Morán, F. J., and López-Jiménez, P. A. 2016. "Performance assessment of OpenFOAM and FLOW-3D in the numerical modeling of a low Reynolds number hydraulic jump." *Environ. Modell. Software*, 80, 322-335.

Bayon, A., and Lopez-Jimenez, P. A. 2015. "Numerical analysis of hydraulic jumps using OpenFOAM." *J. Hydroinf.*, 17(4), 662-678.

Biswas, R., and Strawn, R. C. 1998. "Tetrahedral and hexahedral mesh adaptation for CFD problems." *Appl. Numer. Math.*, 26(1-2), 135-151.

Bélanger, J. B. 1841. "Notes sur l'Hydraulique." *Ecole Royale des Ponts et Chaussées, Paris, France, session*, 1842, 223.

Bennett, N. D., Croke, B. F., Guariso, G., Guillaume, J. H., Hamilton, S. H., Jakeman, A. J., ... and Pierce, S. A. 2013. “Characterising performance of environmental models.” *Environ. Modell. Software*, 40, 1-20.

Blocken, B., and Gualtieri, C. 2012. “Ten iterative steps for model development and evaluation applied to Computational Fluid Dynamics for Environmental Fluid Mechanics.” *Environ. Modell. Software*, 33, 1-22.

Bombardelli, F. A., Meireles, I., and Matos, J. 2011. “Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways.” *Environ. Fluid Mech.*, 11(3), 263-288.

Bradshaw, P. 1997. “Understanding and prediction of turbulent flow—1996.” *Int. J. Heat Fluid Flow*, 18(1), 45-54.

Caishui, H. O. U. 2012. “Three-dimensional numerical analysis of flow pattern in pressure forebay of hydropower station.” *Procedia Engineering*, 28, 128-135.

Caisley, M. E., Bombardelli, F. A., and Garcia, M. H. 1999. “Hydraulic model study of a canoe chute for low-head dams in Illinois.” *Civil Engineering Studies, Hydraulic Engineering Series No-63, University of Illinois at Urbana-Champaign.*

Castillo, L. G., Carrillo, J. M., García, J. T., & Vigueras-Rodríguez, A. 2014. “Numerical simulations and laboratory measurements in hydraulic jumps.” In *Proceedings of the 11th nternational Conference of Hydroinformatics.* New York City, USA.

Castro-Orgaz, O., and Hager, W. H. 2009. “Classical hydraulic jump: Basic flow features.” *J. Hydraul. Res.*, 47(6), 744-754.

Celik, I. B., Ghia, U., and Roache, P. J. 2008. “Procedure for estimation and reporting of uncertainty due to discretization in {CFD} applications.” *Journal of fluids {Engineering-Transactions} of the {ASME}*, 130(7).

Chachereau, Y., and Chanson, H. 2011. “Free-surface fluctuations and turbulence in hydraulic jumps.” *Exp. Therm Fluid Sci.*, 35(6), 896-909.

Chanson, H. 2013. “Hydraulics of aerated flows: qui pro quo?.” *J. Hydraul. Res.*, 51(3), 223-243.

Chanson, H. 2009. “Current knowledge in hydraulic jumps and related phenomena. A survey of experimental results.” *Eur. J. Mech. B Fluids*, 28(2), 191-210.

Chanson, H., and Gualtieri, C. 2008. “Similitude and scale effects of air entrainment in hydraulic jumps.” *J. Hydraul. Res.*, 46(1), 35-44.

Chanson, H. 2006. “Bubble entrainment, spray and splashing at hydraulic jumps.” *J. Zhejiang Univ. Sci. A*, 7(8), 1396-1405.

Chanson, H., and Brattberg, T. 2000. “Experimental study of the air–water shear flow in a hydraulic jump.” *Int. J. Multiphase Flow*, 26(4), 583-607.

Chanson, H., and Montes, J. S. 1995. “Characteristics of undular hydraulic jumps: Experimental apparatus and flow patterns.” *J. Hydraul. Eng.*, 121(2), 129-144.

Cheng, C. K., Tai, Y. C., and Jin, Y. C. 2017. “Particle image velocity measurement and mesh-free method modeling study of forced hydraulic jumps.” *J. Hydraul. Eng.*, 143(9), 04017028.

Chow, V. T. 1959. *Open-channel hydraulics* (Vol. 1). New York, USA: McGraw-Hill Book Company, Inc.

Dong, Z., Wang, J., Vetsch, D. F., Boes, R. M., and Tan, G. 2019. “Numerical Simulation of Air–Water Two-Phase Flow on Stepped Spillways behind X-Shaped Flaring Gate Piers under Very High Unit Discharge.” *Water*, 11(10), 1956.

Falvey, H. T. 1980. “Air-water flow in hydraulic structures.” *NASA STI/Recon Technical Report N, 81*.

Fuentes-Pérez, J. F., Silva, A. T., Tuhtan, J. A., García-Vega, A., Carbonell-Baeza, R., Musall, M., and Kruusmaa, M. 2018. “3D modelling of non-uniform and turbulent flow in vertical slot fishways.” *Environ. Modell Software*, 99, 156-169.

Gualtieri, C., and Chanson, H. 2007. “Experimental analysis of Froude number effect on air entrainment in the hydraulic jump.” *Environ. Fluid Mech.*, 7(3), 217-238.

Hager, W. H. 1992. *Energy dissipators and hydraulic jump* (Vol. 8). Dordrecht, Netherlands: Kluwer Academic Publishers.

Hager, W. H., Bremen, R., and Kawagoshi, N. 1990. “Classical hydraulic jump: length of roller.” *J. Hydraul. Res.*, 28(5), 591-608.

Hager, W. H., and Bremen, R. 1989. “Classical hydraulic jump: sequent depths.” *J. Hydraul. Res.*, 27(5), 565-585.

Heller, V. 2011. “Scale effects in physical hydraulic engineering models.” *J. Hydraul. Res.*, 49(3), 293-306.

Hirsch, C. 2007. *Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics*. UK: Elsevier.

Hirt, C. W., and Nichols, B. D. 1981. “Volume of fluid (VOF) method for the dynamics of free boundaries.” *J. Comput. Phys.*, 39(1), 201-225.

Ho, D. K. H., and Riddette, K. M. 2010. “Application of computational fluid dynamics to evaluate hydraulic performance of spillways in Australia.” *Aust. J. Civ. Eng.*, 6(1), 81-104.

Jesudhas, V., Balachandar, R., Roussinova, V., and Barron, R. 2018. “Turbulence characteristics of classical hydraulic jump using DES.” *J. Hydraul. Eng.*, 144(6), 04018022.

Jesudhas, V., Roussinova, V., Balachandar, R., & Barron, R. 2016. “Submerged hydraulic jump study using DES.” *J. Hydraul. Eng.*, 143(3), 04016091.

von Kármán, T. 1930. “Mechanische Ähnlichkeit und Turbulenz.” In *Proceedings of the 3rd International Congress on Applied Mechanics*. Stockholm, Sweeden.

Keyes, D., Ecer, A., Satofuka, N., Fox, P., and Periaux, J. (2000). *Parallel Computational Fluid Dynamics' 99: Towards Teraflops, Optimization and Novel Formulations*. Amsterdam, The Netherlands: Elsevier.

Kim, J. J., and Baik, J. J. 2004. “A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k–ε turbulence model.” *Atmos. Environ.*, 38(19), 3039-3048.

Kim, S. E., and Boysan, F. 1999. “Application of CFD to environmental flows.” *J. Wind Eng. Ind. Aerodyn.*, 81(1-3), 145-158.

Kirkgöz, M. S., and Ardiçlioğlu, M. 1997. “Velocity profiles of developing and developed open channel flow.” *J. Hydraul. Eng.*, 123(12), 1099-1105.

Langhi, M., and Hosoda, T. 2018. “Three-dimensional unsteady RANS model for hydraulic jumps.” *ISH J. Hydraul. Eng.*, 1-8.

Liu, Z. P., Guo, X. L., Xia, Q. F., Fu, H., Wang, T., and Dong, X. L. 2018. “Experimental and numerical investigation of flow in a newly developed vortexdrop shaft spillway.” *J. Hydraul. Eng.*, 144(5), 04018014.

Liu, M., Rajaratnam, N., & Zhu, D. Z. 2004. “Turbulence structure of hydraulic jumps of low Froude numbers.” *J. Hydraul. Eng.*, 130(6), 511-520.

Lobosco, R. J., Schulz, H. E., and Simões, A. L. A. 2011. *Analysis of two phase flows on stepped spillways. Hydrodynamics-Optimizing Methods and Tools*. Rijeka, Croatia: IntechOpen.

Long, D., Rajaratnam, N., Steffler, P. M., and Smy, P. R. 1991. "Structure of flow in hydraulic jumps." *J. Hydraul. Res.*, 29(2), 207-218.

Ma, J., Oberai, A. A., Lahey, R. T., and Drew, D. A. 2011. "Modeling air entrainment and transport in a hydraulic jump using two-fluid RANS and DES turbulence models." *Heat Mass Transfer*, 47(8), 911.

McCorquodale, J. A., and Khalifa, A. 1983. "Internal flow in hydraulic jumps." *J. Hydraul. Eng.*, 109(5), 684-701.

McDonald, P. W. 1971. "The computation of transonic flow through two-dimensional gas turbine cascades." In *Proceedings of the ASME 1971 International Gas Turbine Conference and Products Show*. Houston, USA.

Mossa, M. 1999. "On the oscillating characteristics of hydraulic jumps." *J. Hydraul. Res.*, 37(4), 541-558.

Mossa, M., and Tolve, U. 1998. "Flow visualization in bubbly two-phase hydraulic jump." *ASME. J. Fluids Eng.*, 120(1): 160–165

Padulano, R., Fecarotta, O., Del Giudice, G., and Carravetta, A. 2017. "Hydraulic design of a USBR Type II stilling basin." *J. Irrig. Drain. Eng.*, 143(5), 04017001.

Peterka, A. J. 1964. *Hydraulic design of stilling basins and energy dissipators*. USA: US Government Printing Office.

Pope, S. B. 2001. *Turbulent flows*. UK: Cambridge University Press.

Resch, F. J., and Leutheusser, E. H. 1972. "Le ressaut hydraulique: mesures de turbulence dans la région diphasique." *La Houille Blanche*, (4), 279-293.

Romagnoli, M., Portapila, M., and Morvan, H. 2009. "Computational simulation of a hydraulic jump" [Simulación computacional del resalto hidráulico]. [In Spanish] *Mecánica Computacional, XXVIII*, 1661-1672.

Sarfaraz, M., and Attari, J. 2011. "Numerical Simulation of Uniform Flow Region over a Steeply Sloping Stepped Spillway." In *Proceedings of the 6th National Congress on Civil Engineering*. Semnan, Iran.

Spalart, P. R. 2000. "Strategies for turbulence modelling and simulations." *Int. J. Heat Fluid Flow*, 21(3), 252-263.

Speziale, C. G., and Thangam, S. 1992. "Analysis of an RNG based turbulence model for separated flows." *Inte. J. Eng. Sci.*, 30(10), 1379-IN4.

Tajabadi, F., Jabbari, E., and Sarkardeh, H. (2017). "Hydrodynamic analysis of flow in USBR stilling basin types I, II, III." *J. Iran. Dam Hydroelec. Powerplant*, 4(13), 43-54.

Teuber, K.; Broacker, T; Bayón, A.; Nützmann, G. and Hinkelmann, R. 2019. "CFD-modelling of free surface flows in closed conduits." *Prog. Comput. Fluid Dyn. an Int. J*, 19(6), 368–380.

Toso, J. W., and Bowers, C. E. 1988. "Extreme pressures in hydraulic-jump stilling basins." *J. Hydraul. Eng.*, 114(8), 829-843.

Valero, D., Viti, N., and Gualtieri, C. 2019. "Numerical Simulation of Hydraulic Jumps. Part 1: Experimental Data for Modelling Performance Assessment." *Water*, 11(1), 36.

Valero, D., and Bung, D. B. 2016. "Sensitivity of turbulent Schmidt number and turbulence model to simulations of jets in crossflow." *Environ. Modell. Software*, 82, 218-228.

Valero, D., and Bung, D. B. 2015. "Hybrid investigations of air transport processes in moderately sloped stepped spillway flows." In *E-proceedings of the 36th IAHR World Congress* (Vol. 28, pp. 1-10). The Hague, The Netherlands.

Viti, N., Valero, D., and Gualtieri, C. 2019. "Numerical simulation of hydraulic jumps. Part 2: Recent results and future outlook." *Water*, 11(1), 28.

Wang, H., and Chanson, H. 2015a. "Experimental study of turbulent fluctuations in hydraulic jumps." *J. Hydraul. Eng.*, 141(7), 04015010.

Wang, H., & Chanson, H. 2015b. "Air entrainment and turbulent fluctuations in hydraulic jumps." *Urban Water J.*, 12(6), 502-518.

Wang, H. 2014. "Turbulence and air entrainment in hydraulic jumps," PhD Thesis. University of Queensland, Australia.

Weller, H. G., Tabor, G., Jasak, H., and Fureby, C. 1998. "A tensorial approach to computational continuum mechanics using object-oriented techniques." *Comput. Phys.*, 12(6), 620-631.

Witt, A., Gulliver, J., and Shen, L. 2015. "Simulating air entrainment and vortex dynamics in a hydraulic jump." *Int. J. Multiphase Flow*, 72, 165-180.

Wu, J. H., Zhou, Y., and Ma, F. 2018. "Air entrainment of hydraulic jump aeration basin." *J. Hydrodyn.*, 30(5), 962-965.

Xiang, M., Cheung, S. C. P., Tu, J. Y., and Zhang, W. H. 2014. "A multi-fluid modelling approach for the air entrainment and internal bubbly flow region in hydraulic jumps." *Ocean Eng.*, 91, 51-63.

Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B., and Speziale, C. G. 1992. "Development of turbulence models for shear flows by a double expansion technique." *Phys. Fluids A* 4(7), 1510-1520.

Zhang, G., Wang, H., and Chanson, H. 2013. "Turbulence and aeration in hydraulic jumps: free-surface fluctuation and integral turbulent scale measurements." *Environ. Fluid Mech.*, 13(2), 189-204.

FIGURE CAPTIONS

- **Fig. 1.** Meshed domain and detail of the refined and coarse mesh blocks. Adapted from Bayón *et al.* (2019).
- **Fig. 2.** Dimensionless free surface profile comparing FLOW-3D, OpenFOAM, Wang & Chanson (2015a), Bakhmeteff & Matzke (1936) and experimental results.
- **Fig. 3.** Velocity analysis: a) Maximum forward velocity decay; b) Maximum backward velocity.
- **Fig. 4.** Velocity profiles along the longitudinal axis in the hydraulic jump roller region. a) FLOW-3D. b) OpenFOAM. c) Experimental.
- **Fig. 5.** Vertical velocity profiles along the longitudinal axis for the physical and the numerical models compared to Kirkgoz & Ardiclioglu (1997). a) Supercritical regime. b) Subcritical regime.
- **Fig. 6.** Pressure analysis. a) Relative pressures along the hydraulic jump longitudinal axis comparing results from FLOW-3D, OpenFOAM, experimental model and Toso & Bowers (1988). b) Distribution of pressure fluctuations P/Pm for FLOW-3D, OpenFOAM, experimental model and Abdul Khader & Elango (1974).

TABLES

Table 1. Summary table. Capability of the results obtained by the CFD models to represent the case study, compared to the experimental and bibliographical results.

Variable	Referred to	FLOW-3D	OpenFOAM	Compared to
Sequent depth	Accuracy	94.2%	94.7%	Exp. (DIP)
		96.4%	96.9%	Exp. (Limnimeters)
		96.4%	96.9%	Exp. (Ultrasound)
		96.5%	97.0%	Hager & Bremen (1989)
Hydraulic jump efficiency		96.1%	96.4%	Exp. (DIP)
		97.7%	98.0%	Exp. (Limnimeters)
		97.7%	98.0%	Exp. (Ultrasound)
		99.0%	99.3%	Hager (1992)
Roller length		89.2%	98.7%	Experimental
		85.9%	97.5%	Hager *et al.*(1990)
		93.3%	94.0%	Wang & Chanson (2015a)
Free surface profile	R^2	0.943	0.961	Exp. (DIP)
		0.991	0.996	Bakhmeteff & Matzke (1936)
		0.956	0.996	Wang & Chanson (2015a)
Maximum velocity decay		0.872	0.868	Experimental
		0.999	0.992	Hager (1992)
Maximum backward velocity		0.858	0.754	Experimental
		0.618	0.928	Hager (1992)
Velocity profiles in the roller		0.984	0.978	Hager (1992)
Subcritical velocity profiles		0.979	0.973	Hager (1992)
Supercritical velocity profiles		0.981	0.903	Kirkgoz & Ardiclioglu (1997)
Pressure along the longitudinal axis		0.995	0.958	Toso & Bowers (1988)