© 3rd IAHR Europe Congress, Book of Proceedings, 2014, Porto - Portugal. ISBN 978-989-96479-2-3

EXPERIMENTAL AND NUMERICAL STUDY OF WATER INTAKES: CASE STUDY OF THE FOZ TUA HYDROPOWER PLANT

INÊS MEIRELES(1), SORAIA SILVA(1) , TERESA VISEU(2) & VITOR SOUSA(3)

(1) *University of Aveiro, Aveiro, Portugal*
imeireles@ua.pt, silva.s@ua.pt

(2) *National Laboratory of Civil Engineering - LNEC, Lisbon, Portugal*
tviseu@lnec.pt

(3) *Technical University of Lisbon - IST, Lisbon, Portugal*
vitor.sousa@ist.utl.pt

Abstract

Studies on the hydraulics of water intakes have been traditionally performed in physical models. In order to validate the use of computational fluid dynamics models to characterize the flow in water intakes in terms of velocity and pressure, a comparative experimental-numerical study was performed, centered in the case study of the Foz Tua pumped-storage hydropower plant. Overall, was observed proximity between numerical and experimental data, predicting that numerical models can be used to simulate flow in water intakes.

Keywords: water intake; experimental study; numerical study; velocity; pressure

1. Introduction

Traditionally, studies of complex flows in hydraulic structures, namely water intakes, have been performed through physical modeling. However, the fast development of computers in terms of memory and speed make possible the use of computational fluid dynamics (CFD) models to simulate hydraulic structures flows in a reasonable period of time. For this reason, the dam engineering community started recently to make use of CFD models, namely in spillways and water intakes (e.g., Higgs and Frizell, 2004; Groeneveld *et al.*, 2007; Ho and Riddette, 2010; Vasquez *et al.*, 2013). However, although physical models are subjected to scale effects and are more expensive and time-consuming than numerical models, the latter are not exempt of errors due to mathematical and numerical approximations. In a society where the resources are scarce, the combination of these tools will contribute to more sustainable studies and optimized solutions. Ultimately, the CFD models can be used for situations already validated.

Although several studies have been published regarding the numerical modeling of horizontal water intakes, namely focused on vorticity (e.g., Suerich-Gulick *et al.*, 2006), pressure (e.g., Groeneveld *et al.*, 2007), velocity (e.g., Bermúdez *et al.*, 2012), sediment transport (e.g., Ruether *et al.*, 2005), or juvenile fish bypass systems (e.g., Khan *et al.*, 2004), a comprehensive numerical study on the hydraulics of water intakes with the comparison of numerical data with field or physical model data is still missing. The present study is focused on the comparison of experimental and numerical results of velocity and pressure at the water intake of Foz Tua pumped-storage hydropower plant with the aim of decreasing this lack of knowledge.

2. Physical model

Foz Tua is a hydro power project located in the Tua river, Portugal. The project includes a double curvature arch dam, with a surface spillway, a bottom outlet and two hydraulic circuits for hydroelectric power production, equipped with turbine-pump groups. These two turbine-pump circuits begin with two vertical water intakes placed in the reservoir, approximately 80 m upstream of the dam body, followed by two parallel tunnels with a diameter of 7.5 m and a power house equipped with the pump-turbines.

The two turbine-pump circuits were tested in a reduced scale model at the National Laboratory of Civil Engineering (LNEC), Lisbon, Portugal (Figure 1). The general model scale of 1/65.79 was selected according to Froude similarity.

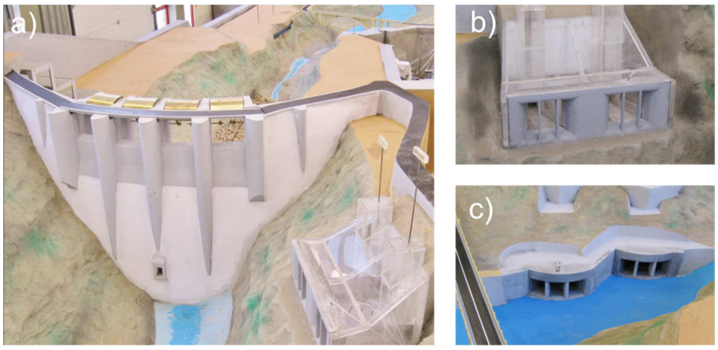
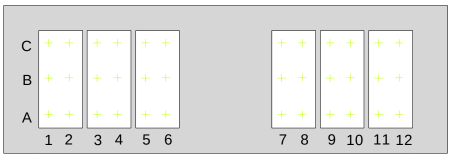
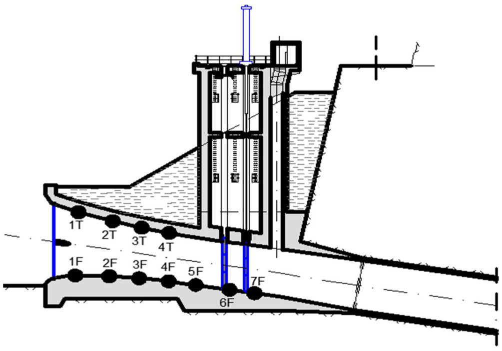


Figure 1. Foz Tua dam scale model: a) dam upstream view and water intake in the reservoir; b) frontal view of the water intake; c) outlet structures.

The experimental tests consisted in measuring velocity and pressure. Velocities through the trashracks were measured in a grid of 36 points in three distinct horizontal planes (Figure 2a), using micro current meters and presenting a measurement error of less than 5%. Piezometric pressure taps were installed in the water intakes to monitor pressures (Figure 2b).





a) b)

Figure 2. Location of the measurement points of: a) velocity; and b) pressure.

An operating condition was tested, corresponding to the Maximum Water Level. The tests were undertaken for a discharge of 2 x 4.42 l/s, which is the scale turbinated maximum flow. More information can be found in Silva (2011).

3. Numerical model

In this study, the simulations were performed with *FLOW-3D*®, a commercial CFD software which allows to model three-dimensional, transient, multi-scale, multi-physis flows (Flow Science, 2011). The code resolves the fluid equations of motion combining the discretization methods of finite differences and finite volumes in a Cartesian grid.

3.1 Governing equations

For this study the governing equations are:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \qquad [1]$$

$$\rho_0 \left[\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \left(\overline{u_i}\,\overline{u_j} \right)}{\partial x_j} \right] = \rho_0 g_i + \mu \frac{\partial^2 \overline{u_i}}{\partial {x_j}^2} - \frac{\partial \overline{p}}{\partial x_i} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \qquad [2]$$

where $\overline{u_i}$ and $\overline{u_j}$ are the time–averaged velocity components in the directions x_i and x_j; ρ_0 is the reference density; t is the time coordinate; g_i is the component of the acceleration of gravity in the direction x_i; μ is the dynamic viscosity; $\overline{p}$ is the time–averaged, modified pressure; and u_i' and u_j' are the fluctuating velocity components in the directions x_i and x_i. $-\rho \overline{u_i' u_j'}$ represent, physically, the transport of momentum due to the turbulent motion, acting as additional stresses on the fluid, being called Reynolds stresses. To close the problem, turbulence was modeled considering the Boussinesq approach and taking into account the standard k - ε turbulence model (Launder and Spalding 1972).

3.2 Boundary conditions

In the present study, the domain where the governing equations are valid is composed by part of the reservoir, the water intake and part of the hydraulic circuit.

To located the free surface, *FLOW-3D*® uses the *TruVOF* method (Hirt and Nichols, 1981), which has three key elements: locating the free surface through a function *F* which defines the volume fraction of each cell occupied by fluid; considering an appropriated transport equation for *F*; and applying boundary conditions to the free surface. This method presents the advantage of congregating low computational memory requirements and high accuracy at a low computational cost (Flow Science, 2011).

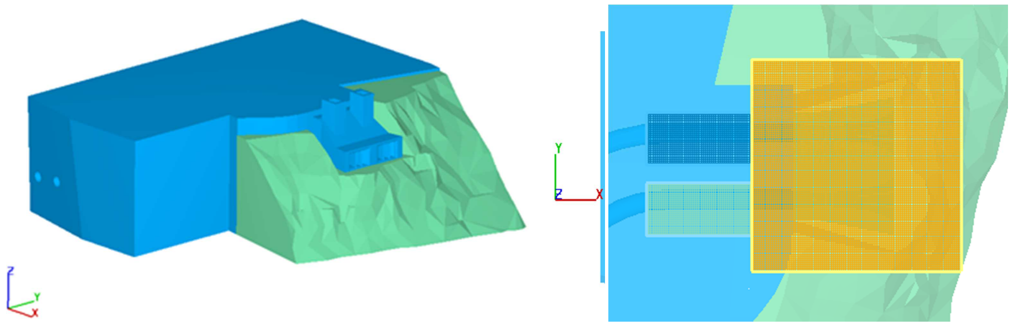
Were also considered the following boundary conditions: i) volumetric flow rate at the exit of the tunnels of the hydraulic circuit; ii) water depth in the reservoir; and iii) rigid wall at the bottom of the reservoir. At the solid surfaces was imposed null normal velocity (Pope, 2000) and used the usual wall functions to take into account turbulence close to the walls (Pope, 2000; Ferziger and Peric, 2002).

3.3 Numerical model implementation

In *FLOW-3D*®, objects can be generated directly by a solid modeler or imported from I-DEAS universal files, ANSYS tetrahedral or surface triangle files, stereolithography (STL) files, or topographical data files (Flow Science, 2011). Grids are automatically generated through the definition of the domain size and number of cells (or cells dimension) in each direction. It is possible to consider nested or linked grid blocks (Flow Science, 2011; Barkhudarov, 2004).

In *FLOW-3D®* grids and objects are generated independently and subsequently combined with the Fractional Area/Volume Obstacle Representation technique (*FAVOR™*). Developed by Hirt and Sicilian (1985), this technique defines the geometry based on the fraction of the cells occupied by the objects.

In the present study, the solid geometry was defined in a CAD file, based on the dimensions of the physical model, and subsequently imported to *FLOW-3D®* in STL format (Figure 3a). As presented in Figure 3b, the grid was composed by three uniform blocks, defined in order to allow focusing on the zone of interest.



a) b)

Figure 3. Solid and grid definition: a) solid; b) grid.

Grid convergence was studied in detail, to assure that the results would be independent of its resolution (Table 1).

Table 1. Characteristics of the grids used in the convergence analysis.

	L (m)	Number of cells (x10^6)
RUN 1	0.050	0.02
RUN 2	0.025	0.13
RUN 3	0.020	0.25
RUN 4	0.015	0.69
RUN 5	0.012	1.36
RUN 6	0.010	2.35

Figure 4 presents an example of the results obtained during grid convergence analysis for the velocities through the trashracks of the water intake. The results for Runs 5 and 6 are virtually identical, showing that grid convergence was attained. This study can, then, be performed considering a grid with the level of refinement of that used in Run 5, allowing to save memory and computation time in comparison to using the grid of Run 6.

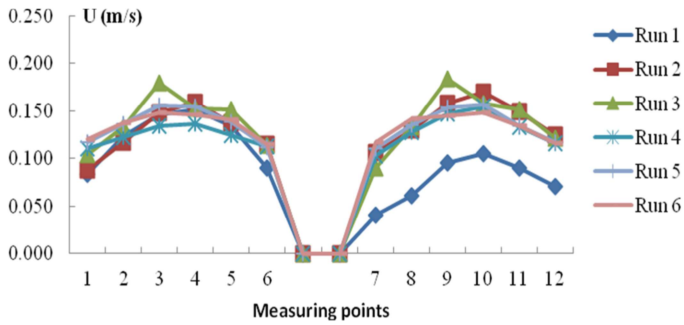


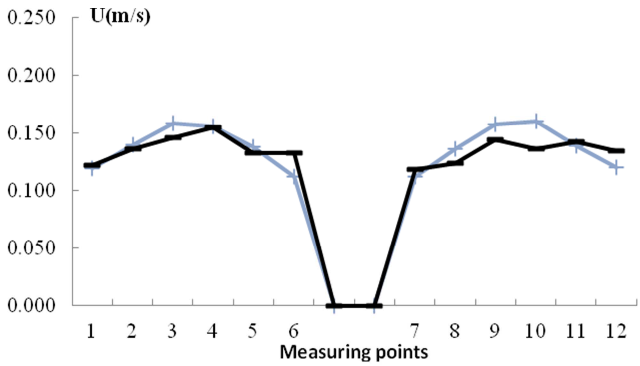
Figure 4. Grid convergence analysis: results from the comparison between velocities through the trashraks of the water intake.

4. Experimental and numerical results

4.1 Velocity

In the physical model, the velocity field captured at the entrance of the water intakes was observed to be practically uniform, with slightly higher values at the center of each water intake. Is observed some symmetry of the results relatively to the vertical and horizontal planes that intercept at the center of each water intake. In the experimental tests were obtained flow velocities up to 0.155 m/s (corresponding to 1.26 m/s at the prototype), which, according to USBR (1987), is bellow the limit for trashracks accessible for cleaning, of 1.50 m/s. The numerical simulations returned a velocity field with similar pattern, although its symmetry was more pronounced. Numerical simulations returned velocities up to 0.161 m/s (1.30 m/s at the prototype).

The averaged relative difference between experimental and numerical velocities is of 7.3% and the absolute maximum is of 18.8%, which are mainly within the expected experimental errors. In addition, the representation of the topography was not completely equal in the physical and numerical models, which can be an additional reason for these differences. It should also be noticed that the relative differences comprise errors from the experimental study (sum of the measuring and instrumentation errors) and from the numerical simulations (due to mathematical and numerical approximations). Overall, the experimental and numerical velocity fields are similar (Figure 4), allowing to conclude that the numerical model was able to capture the behavior of the flow velocity, both qualitatively and quantitatively.



a)

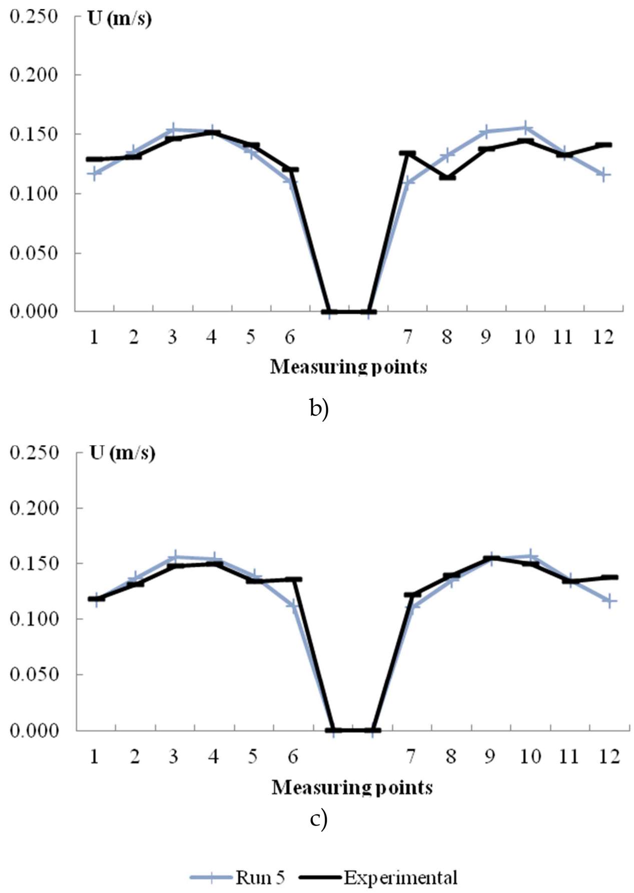


Figure 4. Comparison between experimental and numerical velocities through the trashracks at different planes: a) plane A; b) plane B; and c) plane C.

4.2 Pressure

According to USBR (1987), the form of the water intake should be streamlined to provide smooth, gradual changes in the flow, minimizing head losses and avoiding zones where cavitation pressures can develop. From both, the experimental and the numerical studies, the form of the water intake is observed to be appropriate, since the pressure field presents gradual changes with depth and are not observed negative values (Table 2).

Experimental and numerical results of pressure head, p/γ, obtained in the centerline of the left water intake are observed to be close, with relative differences up to 16% (Table 2). These results are in accordance to Groeneveld *et al.* (2007), who obtained a reasonably good agreement of average pressure magnitudes in a comparative study between pressure results obtained with pressure transducers in the physical model of Cabinet Gorge dam and numerical simulations with *FLOW-3D®*.

Table 2. Comparison between experimental and numerical pressure head values in the water intake.

Pressure tap	$p/\gamma_{experimental}$ (m)	$p/\gamma_{numerical}$ (m)	Rel. Diff. (%)
1F	0.436	0.391	10.3
2F	0.435	0.393	9.6
3F	0.436	0.390	10.5
4F	0.440	0.390	11.4
5F	0.447	0.391	12.5
6F	0.455	0.393	13.6
7F	0.463	0.392	15.3
1T	0.291	0.338	16.3
2T	0.310	0.338	9.0
3T	0.321	0.349	8.8
4T	0.331	0.343	3.7

4.3 Volumetric flow rate

To verify that the experimental and numerical velocity data provided the right volumetric flow rate, the results of velocity through the trashracks were integrated numerically, using the composite midpoint rule (Burden and Faires, 2011). The relative difference among experimental and numerical values of the volumetric flow rate obtained from the numerical integration was less than 0.5%, which constitutes an excellent agreement.

5. Conclusions

The use of CFD models is attractive, since, comparing to physical models, they allow to obtain results in a minor period of time and at a reduced cost. However, results are subject to error due to mathematical and numerical approximations. For this reason, CFD models should be validated for each flow situation. In this regard, the proximity between experimental and numerical data of velocity and pressure for the water intake of Foz Tua dam contributed to the validation of the numerical model *FLOW-3D®* to simulate flows in water intakes.

Acknowledgments

The authors would like to thank *EDP - Energias de Portugal, S.A.* for allowing the results of this study to be published.

References

Barkhudarov, M. R. 2004. 'Multi-Block Gridding Technique for Flow-3D', Flow Science Technical Notes, Flow Science, Inc., TN59.

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', *Environmental Fluid Mechanics*, 11, 3, 263-288.

Burden, R. L., and Faires, J. D. 2011. 'Numerical Analysis.' Ninth Edition, Brooks-Cole Publishing.

Ferziger, J. and Peric, M. 2002. 'Computational methods for fluid dynamics', Springer.

Flow Science 2011. '*FLOW-3D®* User's Manual, Version 10', Los Alamos, New Mexico, USA.

Groeneveld, J., Sweeney, P., Mannheim, C., Simonsen, C., Fry, S. and Moen, K. 2007. 'Comparison of intake pressures in physical and numerical models of the Cabinet Gorge dam tunnel', Waterpower XV.

Higgs, J., and Frizell, K. W. 2004. 'Investigation of the Lake Plant Pump Station - Lower Colorado River Authority', Hydraulic Laboratory Report HL-2004-02, Denver Technical Center, Bureau of Reclamation, United States Department of the Interior, Denver, Colorado, December, 2004.

Hirt, C. W. and Nichols, B. D. (1981). 'Volume of Fluid (VOF) method for the dynamics of free boundaries', *Journal of Computational Physics* 39, 1, 201-225.

Hirt, C. W. and Silician, J. M. 1985. 'A porosity technique for the definition of obstacles in rectangular cell meshes, Proc. 4h Int. Conf. Ship Hydro., National Academy of Science, Washington, DC, USA.

Ho, D. K. H. and Riddette, K. M. 2010. 'Application of computational fluid dynamics to evaluate hydraulic performance of spillways in Australia', *Australian Journal of Civil Engineering*, 6, 1, 81-104.

Launder, B. E., and Spalding, D. B. 1972. 'Lectures in Mathematical Models of Turbulence', Academic Press.

Pope, S. B. 2000. 'Turbulent Flows', Cambridge University Press, UK.

Silva, S. 2012. 'Estudo numérico-experimental da tomada de água da barragem de Foz Tua', M.Sc. thesis, University of Aveiro, Portugal (in Portuguese).

USBR 1987. 'Design of small dams', U.S. Government Printing Office, Washington, D.C, USA.

Vasquez, J., Hurtig, K. and Hughes, B. 2013. 'Computational fluid dynamics (CFD) modeling of run-of-river intakes', Proc. Hydrovision 2013, Denver, USA, July.