Jun 17th, 12:00 AM - Jun 20th, 12:00 AM

Physical Modeling and CFD Comparison: Case Study of a Hydro-Combined Power Station in Spillway Mode

Gonzalo Duró

Mariano De Dios

Alfredo López

Sergio O. Liscia

PHYSICAL MODELING AND CFD COMPARISON: CASE STUDY OF A HYDRO-COMBINED POWER STATION IN SPILLWAY MODE

Gonzalo DURÓ
Hydromechanics Laboratory, National University of La Plata, Argentina, gzduro@gmail.com
Mariano DE DIOS and Alfredo LÓPEZ
Hydromechanics Laboratory, Argentina, dediosmariano@gmail.com, lopito.82@gmail.com
Hydromechanics Laboratory Director: Sergio O. LISCIA

ABSTRACT: This study presents comparisons between the results of a commercial CFD code and physical model measurements. The case study is a hydro-combined power station operating in spillway mode for a given scenario. Two turbulence models and two scales are implemented to identify the capabilities and limitations of each approach and to determine the selection criteria for CFD modeling for this kind of structure. The main flow characteristics are considered for analysis, but the focus is on a fluctuating frequency phenomenon for accurate quantitative comparisons. Acceptable representations of the general hydraulic functioning are found in all approaches, according to physical modeling. The k-ε RNG, and LES models give good representation of the discharge flow, mean water depths, and mean pressures for engineering purposes. The k-ε RNG is not able to characterize fluctuating phenomena at a model scale but does at a prototype scale. The LES is capable of identifying the dominant frequency at both prototype and model scales. A prototype-scale approach is recommended for the numerical modeling to obtain a better representation of fluctuating pressures for both turbulence models, with the complement of physical modeling for the ultimate design of the hydraulic structures.

Keywords: CFD validation, hydro-combined, k-ε RNG, LES, pressure spectrum

INTRODUCTION

In the last decades, numerical simulation of three-dimensional flow patterns has become an appealing tool for the representation of the particular dynamics induced by different hydraulic structures, e.g., power station intakes (KHAN et al. 2004), pump intakes (LI et al. 2004), spillways (JOHNSON and SAVAGE 2006), and breach dam breaks (LAROCQUE et al. 2013). In addition, CFD (Computational Fluid Dynamics) has also been applied for fundamental physics research, e.g., ADRIAN (2007) summarized developments through direct numerical simulation and particle image velocimetry of hairpin vortex organization and packet formation. From an engineering perspective, CFD is especially attractive for hydraulic design due to its flexibility in simulating alternative geometries and performing sensitivity analyses, visualization capabilities, modeling large structures or areas, and low costs (once validated) compared to undertaking physical modeling (DEWALS 2013).

On the other hand, physical modeling offers different characteristics to derive adequate hydraulic design and gain insight into the hydrodynamics (NOVAK 2010). The complexity of prototype flows is represented if scale factors are adequately chosen. However, design, building and operation of physical models may take long periods of time. Moreover, flow visualization can be difficult, while non-intrusive and accurate measurement of variables requires care, methodology and appropriate instrumentation.

Careful interpretation and critical analysis should be exercised in both numerical and physical approaches, combined with result validation, in order to use them with confidence when dealing with hydraulic design changes.

The present work aims at, first, introducing the flow characteristics of a hydro-combined power station in spillway mode with a remarkably high discharge capacity, and second, presenting numerical results in contrast with experimental data to provide insight into the capabilities of a commercial CFD code to represent the main hydraulic variables, i.e., discharge, mean pressures, water levels, and vortex shedding frequency. As a consequence, the study intends to contribute to the literature of CFD validations in the case of a complex hydraulic structure and the analysis of prototype-scale CFD modeling results for two turbulence models, and provide insights to identifying the most convenient approaches when facing hydraulic design.

The hydraulic structure in this study is part of a very challenging project that involves the modification of five existing bays in the Aña Cuá spillway, located in the Paraná River, to generate 273 MW rather than freely discharge as at present. Each unit of the powerhouse will be able to operate either as a turbine or as a spillway, thanks to the operation of a second tainter gate.

METHODOLOGY

The study had a hybrid approach, which considered the results of both numerical and physical modeling of the structure in question to characterize its main hydraulic functioning. This composite modeling allowed for further comparisons between these tools to identify respective advantages and drawbacks.

First, an analysis was made at model scale comparing the results of the k-ε RNG (Renormalization-Group) and the LES (Large Eddy Simulation) turbulence models with experimental measurements, taking into consideration the general hydraulic behavior, discharge capacity, water levels in the middle longitudinal profile, mean pressures at fixed locations, and dominant pressure fluctuation frequency at the discharge canal. This last item can be easily identified, particularly at locations 2 and 4 (Figure 4), allowing either of them to be independently studied. Point 3 was not considered due to its more complex pressure spectrum, in which the dominant frequency was not as clearly identifiable as in the aforementioned points.

Second, prototype-scale simulations with both turbulence models were compared with up-scaled experimental data: the discharge and the dominant pressure fluctuation at the discharge canal nose.

The scenario under study comprised a combination of an extraordinary flood event and the minimum reservoir level, which is the worst condition regarding the potential formation of a hydraulic jump over the discharge canal, as identified in both the physical model and the numerical tests. A single unit was analyzed, functioning between two other operating units so that a symmetric approaching flow in the reservoir could be assumed. Finally, one preliminary hydraulic design was studied through both physical and numerical modeling.

Physical modeling

The experimental study was conducted at the National University of La Plata. The physical model (Figure 1) had a 1:40 scale. The power station model had a length of 2.63 meters from the spillway piers to the end of the discharge canal above the turbine (Figure 4), and it was connected downstream and upstream to 0.475 meters wide flumes.

Figure 1 – Physical scale model (left). Upstream flume and point gauge (right)

The water levels were regulated by means of a bell-mouth spillway situated in a cylindrical basin (Figure 1) and a sluice gate, at the upstream and downstream flume extremes, respectively. The discharge was measured from a V-shaped sharp-crested weir with an accuracy estimated to be 4%. The water levels were measured with a point gauge with a vertical accuracy of 0.1 millimeters. The pressure was sampled with a pressure transducer during 200 seconds at 100 Hz.

CFD modeling

The CFD code applied to this study was FLOW-3D™ v9.4.5, developed by FlowScience Inc., which numerically solves the Navier-Stokes equations using the k-ε RNG model or the LES technique and a Smagorinsky subgrid-scale model. The free surface is represented through the Volume-Of-Fluid method (HIRT and NICHOLS 1981). Solid boundaries are defined by the FAVOR® method, the accuracy of which depends on the cell size chosen. A third-order advection method is used to approximate the solutions. Four simulations were carried out to compare results between the two different turbulence models at model and prototype scales.

The CFD modeled geometry (Figure 2) was analogous to the physical model structure and flumes. It had uniform roughness coefficients of 0.0001 meters at model scale and 0.001 meters at prototype scale, assuming for the latter a smooth concrete surface. The domain was divided into three blocks for meshing purposes to optimize the simulation run time: the upstream flume representing the reservoir, the power station, and the downstream flume representing the tail water. The mesh blocks had uniform cubic cells, with sides at model scale that were 0.0125 meters for the flumes and 0.00625 meters for the power station, equaling 1/27 and 1/54 of the total energy head over the spillway crest, respectively. The inlet and outlet boundary conditions were set up as stagnation pressures so that the total energy head at the reservoir and the river could be properly represented.

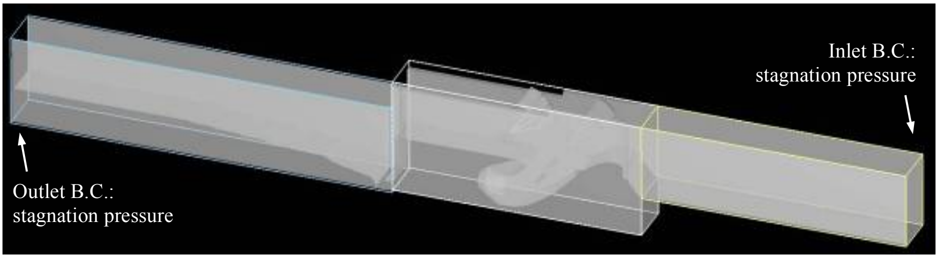

Figure 2 – Mesh blocks for the modeled domain with the boundary conditions

Preliminary simulations with coarse meshes were conducted to reach the steady flow condition. At a later stage, finer meshes were used, which were more demanding in terms of computing time and data storage, to finally arrive at the previously described (Table 1). It is worth mentioning that, in this process, grid independence was achieved in terms of discharge capacity with both turbulence models, i.e., the same volume flow rate, reported in Table 1, was recorded between the final mesh (0.00625 m high cells) and the previous one (0.0125 m high cells). In addition, mass conservation was verified between mesh blocks, showing a maximum discharge difference of 0.2% for all the final simulations. The turbulent mixing length equals 7% of the water depth at the spillway crest for k-ε RNG.

RESULTS

The general behavior predicted by CFD simulations was in agreement with the one observed in the physical model. According to the visual inspection, the flow along the power station was straight until the tailwater, even over the turbine intake and the discharge canal. The flow regime remained supercritical after the spillway crest and no major perturbations were observed along the power station.

Free surface, discharge, and mean pressures

The free surface simulated by the k-ε RNG model appeared smoother than the one obtained by the LES model (Figure 3), and both were able to predict two cross waves arising over the turbine intake. However, only the LES model predicted an oscillating free surface over the discharge canal and small fluctuating diagonal waves against the walls that moved downstream, a phenomenon also confirmed by observation in the physical model.

The discharges obtained by numerical simulations at the model scale were higher than the physical model measured discharge, namely, 13% for the k-ε RNG model and 10% for LES. Down-scaling the discharges from the prototype-scale simulations through Froude's similarity law showed a discharge increase of 2.1% compared to the model-scale numerical results.

There was a fair general agreement between the water levels in the physical model and those obtained with the CFD simulations (Figure 4). However, both numerical models presented higher mean temporal values than the experimental, which can be clearly appreciated over the spillway crest. Over the discharge canal (points 4-10), temporal oscillations of the water level were observed, but as they could not be measured with the point gauge, the applied criterion to make quantification possible was to adopt the highest level during periods of 30 seconds at each point. As a result, the actual mean temporal values over the canal are slightly (approximately 1 cm) below the measured ones, plotted in Figure 4.

The analysis of mean pressures at different fixed locations in the physical model showed that CFD simulations overestimated the experimental data by approximately 13%, which is consistent with the results obtained for the discharges.

Fluctuating phenomenon: vortex shedding

A high shear layer was located between the stream flowing over the turbine intake and the recirculating water near the turbine bulb (blue velocities in Figure 3). The difference in velocity magnitudes between these two regions was significant, and as a consequence, the arising instability led to the formation of vortices. The employed turbulence models yielded divergent results in this respect: the LES wass able to simulate an oscillating free surface and fluctuating pressure and velocity fields (Figure 5), while the k-ε RNG model failed to represent any of these phenomena.

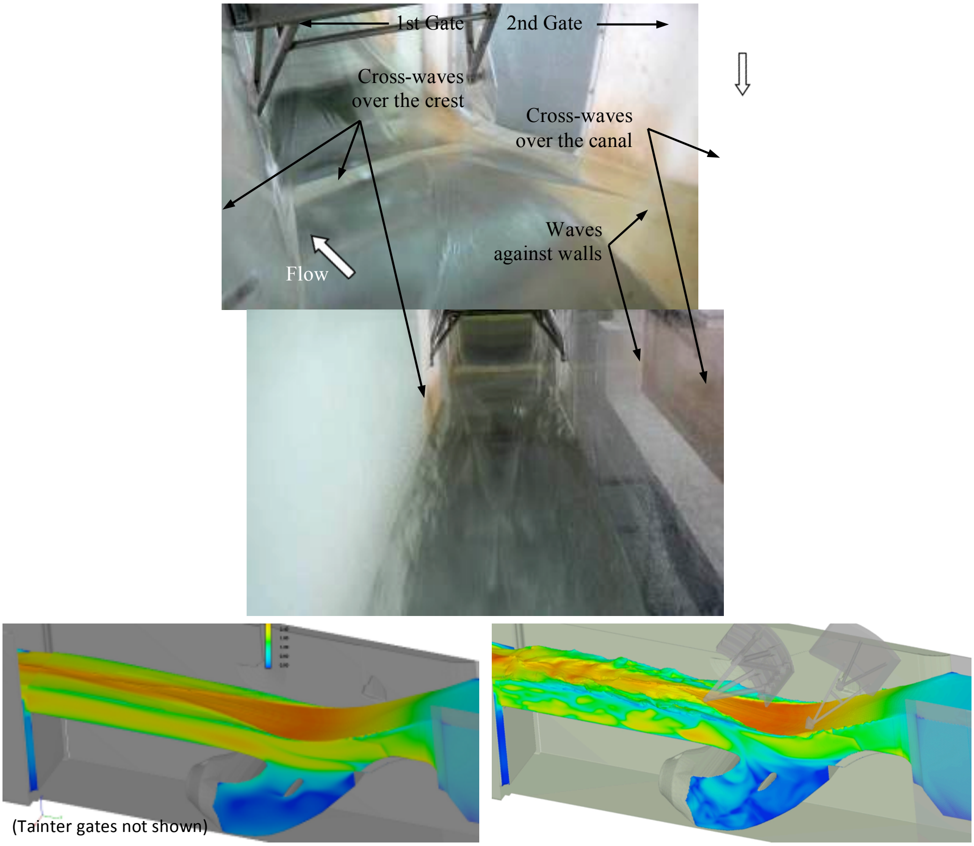

Figure 3 – Free surface views. Bottom left: k-ε RNG model. Bottom right: LES.

Only the frequency of the fluctuations was analyzed, neglecting the amplitude, since the magnitude of the latter was within the pressure transducer error, i.e., 0.02 meters, so no conclusions were drawn in that respect. The pressure acquisition frequency was 100 Hz, right above the Nyquist frequency (GRENANDER 1959), to avoid aliasing, even at the highest considered frequency of analysis, which was 14 Hz (Figure 6).

Table 1 – Discharge flows, simulations characteristics, pressure sampling characteristics

	CFD - Model scale		Physical model	CFD - Prototype scale	
Turbulence model	k-ε RNG	LES	N/A	k-ε RNG	LES
Computing time for 30sec [hr]	142	94	N/A	13	12
Cell size (cube height) [m]	0.00625	0.00625	N/A	0.25	0.25
Discharge (Q_i) at model scale [m³/s]	0.159	0.155	0.141	0.162*	0.158*
Difference: $(Q_i-Q_{PM})*100/Q_{PM}$	12.8%	9.9%	0%	14.9%	12.1%
Discharge at prototype scale [m³/s]	1609*	1568*	1431*	1635	1598
Pressure sampling frequency [Hz]	100	100	100	100	100
Pressure sampling time [sec.]	30	15	200	20	20

*Calculated from Froude's similarity law

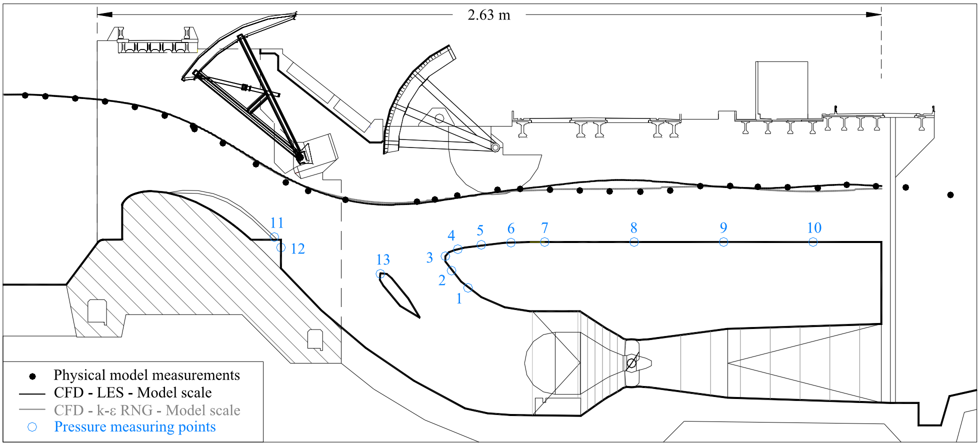

Figure 4 – Water levels: physical model (maximum values) and CFD results (mean values)

Table 2 – Pressure measurements and CFD results. Non-dimensional values: pressure relative to the total energy head over the spillway crest

Pressure point	1	2	3	4	5	6	7	8	9	10	11	12	13
Measured	1.03	0.89	0.92	0.47	0.29	0.37	0.41	0.48	0.47	0.44	0.88	0.78	0.90
k-ε Model scale	1.19	1.02	1.03	0.54	0.40	0.36	0.44	0.53	0.55	0.52	0.89	0.81	1.19
LES Model scale	1.21	1.09	1.05	0.49	0.39	0.39	0.47	0.56	0.58	0.54	0.81	0.84	1.21

A fast Fourier transform analysis of the pressures at point 2 (Figure 4), performed with the software Origin, revealed that there was a dominant frequency of 4.4 Hz in the physical model, which was higher than the 3.9 Hz predicted by the LES model (Figure 6). The mesh discretization, which might not be sufficiently fine to represent this phenomenon more accurately, may have been the cause of this discrepancy, but further research is needed to prove this hypothesis. Another source of error may be the sampling time disparity: 15 s for the numerical tests and 200 s for the physical model.

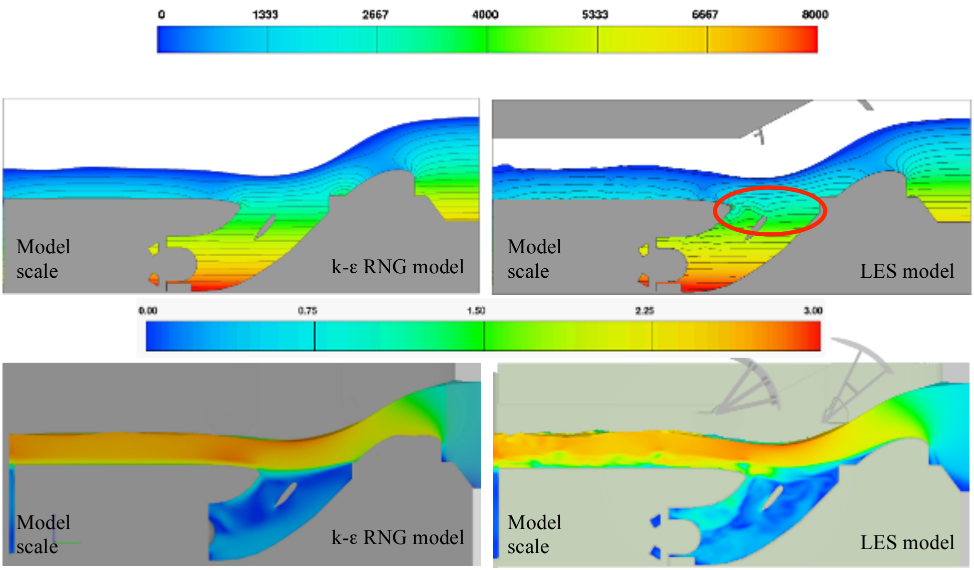

Figure 5 – Instantaneous pressures [Pa] and velocities [m/s] at model scale (bay center)

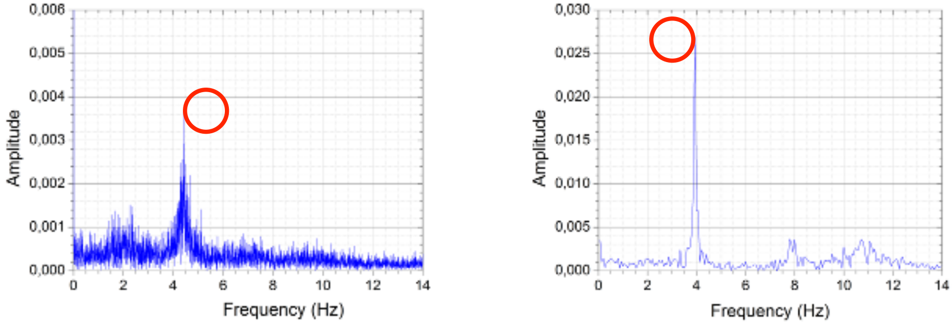

Figure 6 – Energy spectra of pressure at point 2. Left: physical model. Right: LES model

Prototype Scale Results

The simulations performed at prototype scale showed similar general hydraulic behaviors to the model-scale numerical results (Figure 7). However, a 2.1% discharge increase was observed with both turbulence models. The K-ε RNG model and the LES predicted dominant frequencies of 0.70 Hz and 0.65 Hz, respectively (Figure 8). Corresponding Strouhal numbers of 0.22 and 0.21 did not differ substantially from the physical model value of 0.23. To compute the Strouhal numbers (St = f*L/V) the characteristic length and velocity were considered to be the water depth and the average velocity at the middle of the spillway step edge, respectively.

It is worth mentioning that the LES represented the dominant fluctuating phenomenon in spite of the high Reynolds numbers, which might impose limitations, e.g., to represent the boundary layer. These Reynolds numbers were $4.4*10^5$ and $1.1*10^8$ for the model and prototype scales respectively, considering the same location used for the Strouhal number computation.

In light of the fact that the k-ε RNG model with cell heights of 0.50 m in the powerhouse mesh block (preliminary coarser mesh used to accelerate flow stabilization) predicted a dominant frequency of 0.60 Hz, it is deemed likely that coarser meshes would give rise to lower dominant frequencies than the actual ones. This hypothesis would also explain the frequency difference shown in Figure 6 for the LES results at model scale.

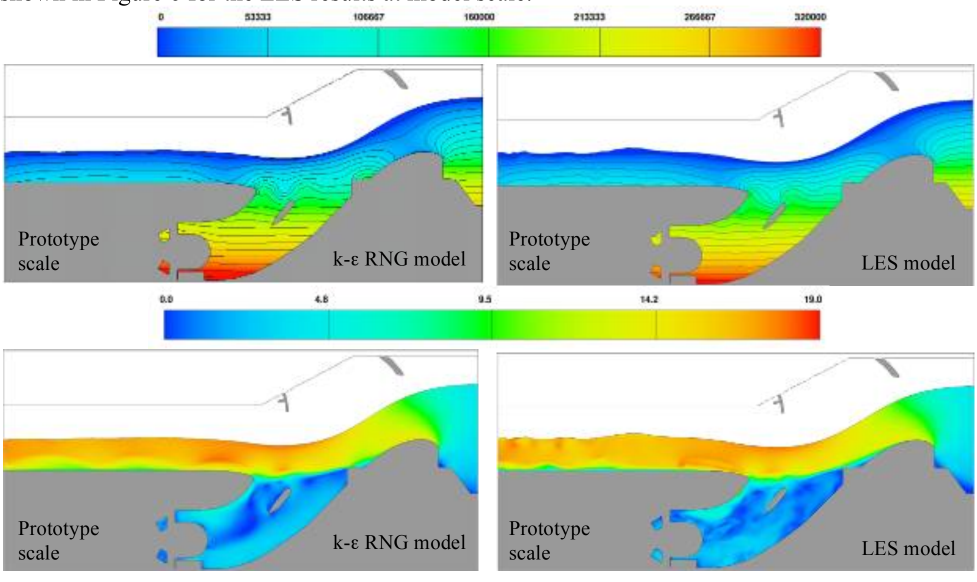

Figure 7 – Instantaneous pressures [Pa] and velocities [m/s] at prototype scale (bay center)

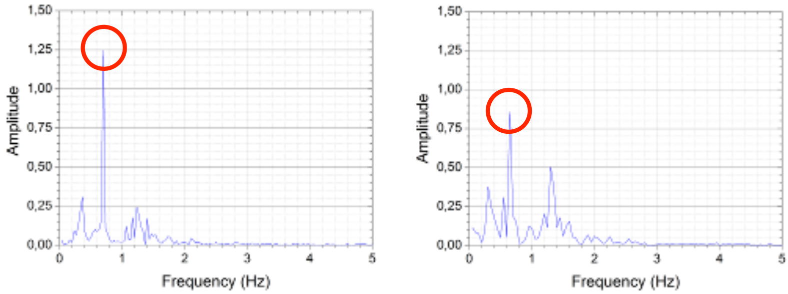

Figure 8 – Energy spectra of pressure in point 2 at prototype scale. Left: k-ε RNG. Right: LES

SUMMARY AND CONCLUSIONS

CFD simulations with both k-ε RNG and LES turbulence models at model and prototype scales provided acceptable representations of the general hydraulic behavior of the power station operating in spillway mode. Discharges predicted by CFD simulations were slightly higher than those measured with the physical model, namely, 13% for k-ε RNG model and 10% for the LES, a trend which holds even considering the estimated measurement error of 4% for the physical model. When numerically modeling at prototype scale, the discharge increased by 2.1% compared to the model scale, irrespective of the turbulence model applied. The analysis of mean pressures and water levels at model scale showed consistency with the aforementioned discharge increase, since higher values than the empirical ones were observed, especially the pressures with an average 13% rise. The presence of cross waves can be predicted in both turbulence models but the inability to empirically quantify mean water levels prevented the realization of more accurate comparisons.

The k-ε RNG model failed to predict fluctuating phenomena at the model scale, but did at the prototype scale regarding the dominant frequency. This discrepancy should be the subject of further research. Additionally, there is some evidence to consider that finer meshes lead to more accurate representation of fluctuating phenomena in terms of frequency, or from another perspective, that coarser meshes give rise to lower frequencies than the actual ones.

The LES closure model predicted the dominant fluctuating frequency fairly well at both model and prototype scales, in terms of pressure and water levels. However, this study cannot guarantee the accuracy of the phenomenon amplitude, so necessary precautions should be taken.

When dealing with design optimization processes in the engineering practice, the accuracy of both turbulence models seem fair enough for this case study, e.g., to suggest a freeboard from the free surface to the beams or to verify the ability of the structure to discharge without a flow regime change. In particular, the k-ε RNG model was good at representing mean flow characteristics, whereas the LES also helped to understand with more detail the physics of the flow, even at small scales. A prototype scale approach was found to be a better representation of fluctuating pressures than the model-scale approach for both turbulence models, and thus it is recommended. Finally, for a complete cost-effective and precise hydraulic design, composite modeling is also recommended considering the aforementioned limitations of all the presented numerical approaches, which still need further development and higher degrees of accuracy to define the ultimate design of the structures.

ACKNOWLEDGMENTS

The authors of this paper would like to acknowledge the support of Entidad Binacional Yacyretá, the National University of Misiones and MWH. The contributions of Eng. Sergio Oscar Liscia, Ph.D. Juan Manuel Galíndez, Ph.D. Raúl Antonio Lopardo and Mrs. Claire Taylor are much appreciated.

REFERENCES

ADRIAN R. J. (2007). "Hairpin vortex organization in wall turbulence." *Phys. Fluids* 19(4), 041301.

DEWALS B., ARCHAMBEAU P., RULOT F., PIROTTON M. and ERPICUM S. (2013). "Physical and Numerical Modelling in Low-Head Structures Design." *Proc. International Workshop on Hydraulic Design of Low-Head Structures, Aachen, Germany, Bundesanstalt für Wasserbau Publ., D.B. BUNG and S. PAGLIARA Editors, pp.11-30.*

GRENANDER, U. (1959). Probability and Statistics: The Harald Cramér Volume. Wiley.

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.

JOHNSON M. C. and SAVAGE B. M. (2006). "Physical and numerical comparison of flow over ogee spillway in the presence of tailwater." *J. Hydraulic Eng*. 132(12): 1353–1357.

KHAN L.A., WICKLEIN E.A., RASHID M., EBNER L.L. and RICHARDS N.A. (2004). "Computational fluid dynamics modeling of turbine intake hydraulics at a hydropower plant." *Journal of Hydraulic Research*, 42:1, 61-69

LAROCQUE L.A., IMRAN J. and CHAUDHRY M. (2013). "3D numerical simulation of partial breach dam-break flow using the LES and k–ϵ turbulence models." *Jl of Hydraulic Research*, 51:2, 145-157

LI S., LAI Y., WEBER L., MATOS SILVA J. and PATEL V.C. (2004). "Validation of a three-dimensional numerical model for water-pump intakes." *Journal of Hydraulic Research*, 42:3, 282-292

NOVAK P., GUINOT V., JEFFREY A. and REEVE D.E. (2010). "Hydraulic modelling - An introduction." Spon Press, London and New York, ISBN 978-0-419-25010-4, 616 pp.