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

EXPERIMENTAL AND NUMERICAL MODELING OF FLOW IN A STILLING BASIN

J.C.C. Amorim(1), R.C.R. Amante(2) & V.D. Barbosa(3)

(1) *Military Institute of Engineering, Rio de Janeiro, Brazil*
jcamorim@ime.eb.br

(2) *Furnas - Eletrobras, Rio de Janeiro, Brazil*
renatacr@furnas.com.br

(3) *Military Institute of Engineering, Rio de Janeiro, Brazil*
dobrochinski_7@hotmail.com

ABSTRACT

This paper presents a study on hydraulic jump stilling basin by using physical model, representing the basin of Porto Colombia Hydropower Plant and numerical modeling. The same situations reproduced in physical model were tested and simulated in a numerical model, and the data of pressure, velocity and water levels from these models were compared. The behavior of the flow inside the stilling basin was well represented by the numerical model and there was a good agreement of their results with the experimental model, concluding the viability of this type of modeling for optimization of stilling basins.

Keywords: stilling basin, turbulence, spillway, hydraulic jump, experimental and numerical model

1. INTRODUCTION

One of dissipative structures often used in a large hydraulic structure is the stilling basin by hydraulic jump. Peterka (1984) conducted a research project in which various types of stilling basins and energy dissipators under various operating conditions were studied in a two-dimensional physical model; however, these studies do not allow the knowledge of the dynamics of the flow and the dissipation energy, important for the optimization study.

Historically, the studies of the performance of the hydraulic jump stilling basins were carried out with physical models. But the necessary time for the construction and the realization of the tests is, sometimes, the limiting factor off the search for the optimization of the projects and the observation of all structure operation conditions. Add to that, the difficulties for the modification of the structures geometric details, that may raise the budget and the time spent on this study.

During the last years, with the continuous development of the computational resources, the numerical models for hydraulic jump predictions have motivated various researchers. The versatility of the numerical models, even if the model is not to be used in the final determination of the best geometry, differently of the physical models, turn the numerical models into valuable tools that can easily be used in the adjustment of various design details and in geometry modifications.

The USBR – United States Bureau of Reclamation developed The Water Resources Research Laboratory (Higgs, 1996). The paper, “Type II and Type III Stilling Basin Modifications Computational Fluid Dynamic Model Study” was published, as a result of this project. This work describes the studies realized in the stilling basin of the Ridgeway Dam, which is a USBR Type II basin. Tests with physical model and simulation with the numerical model were compared, to study the return flows that may carry abrasive material to the inner stilling basin.

Cook et al (2002) propose a free-surface computational fluid dynamics modeling of a spillway and tailrace to study the Dalles project. Comparisons were performed between physical model (Preslan & Wilhelms, 2001) and CFD model results.

The present work presents a numerical and experimental study of the turbulent flow in a hydraulic jump stilling basin. In its development, a physical model and a numerical model based on CFD - Computational Fluid Dynamics – techniques that describe the behavior of turbulent free surface flow were used.

The objective of the study is to determine the characteristics of the turbulent flow in stilling basins, aiming to obtain the interaction between the physical model and the numerical one, through the comparison of the data obtained. In addition, the behavior of the flow inside of a stilling basin is described, focusing the free surface profile, the predictions of the velocities and of the instantaneous and mean pressures in determined points of the flow, and the hydraulic efficiency, using the CDF modeling techniques.

The commercially available CFD model Flow-3D software that was developed by the Flow Science was applied. This is rather satisfactory software for the solutions of equations that represent the free surface turbulent flow.

The physical model used, constructed in the Experimental Hydraulics Laboratory of Furnas S.A. (LAHE), reproduce the geometry of the Hydropower Plant (HPP) of Porto Colombia. The three-dimensional model has a geometric scale of 1:100.

Aligning the physical and the numerical modeling, an important study of all operations conditions is possible; bringing then, a great facility for the study of other structures configurations and, a wide knowledge of the fluid dynamics..

2. EXPERIMENTAL MODEL

The Creager profile spillway of the Porto Colombia HPP has nine bays with radial type gates. Its discharge capacity is of 16,000 m^3/s. The geometry of the basin and the spillway profile are seen in Figure 1, which represents a longitudinal section.

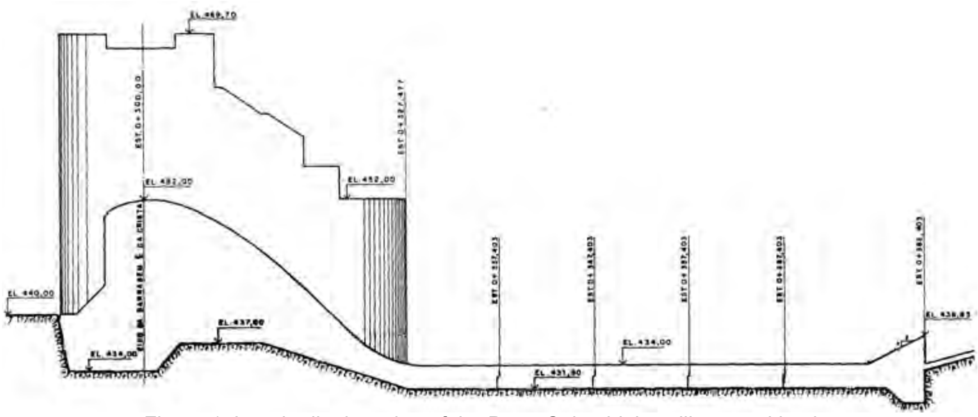

Figure 1. Longitudinal section of the Porto Colombia's spillway and basin.

In this paper an experimental model, also called physical or scale models, was tested and quantities such as pressure, velocity and water level were measured and used for validating the numerical model simulations.

The 1:100 scale model (Figure 2) has a surface area of 120 m2 and represents the main structures of the hydro station plant, and also the bathymetric conditions of the region, in a range of approximately 550 m upstream and 550 m downstream from the axis of the dam with approximately 1000 m wide. The structures reproduced in the model, reproduce part of the dam, the spillway, the stilling basin, the intake and the tailrace.

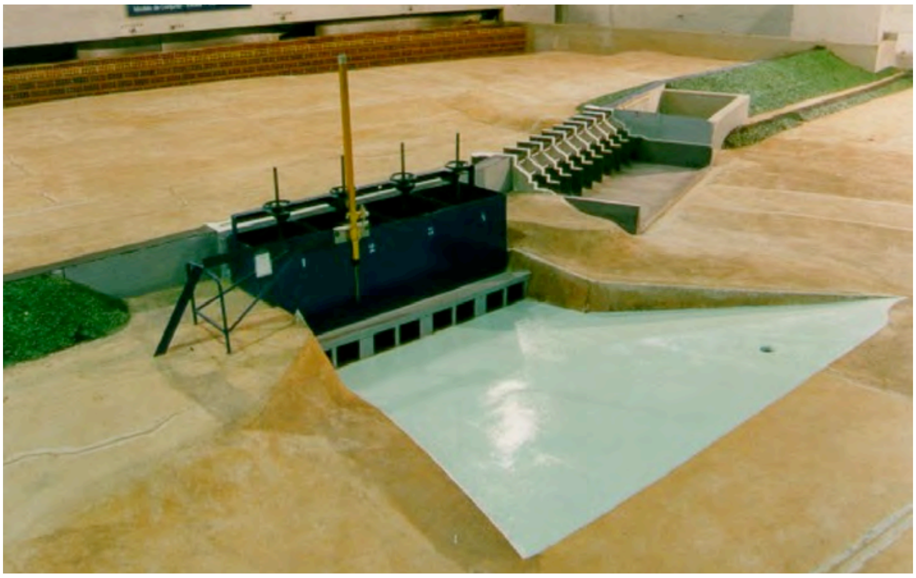

Figure 2. 3D Scale Model of the Porto Colombia Hydropower Plant

There were measured the instantaneous pressures on the bed of the stilling basin, with surface-mounted differential transducers. The pressure measurements were obtained at the first right bay, at the first left one and at the central one. These measurements were done simultaneously at all positions. The measurement points are presented in Figure 3.

The mean velocities at the points C1, C3, C5, C7, C8, and C9 were measured. The two last points are located outside the basin. The water levels at the longitudinal section of the basin were measured, corresponding to central bay axis.

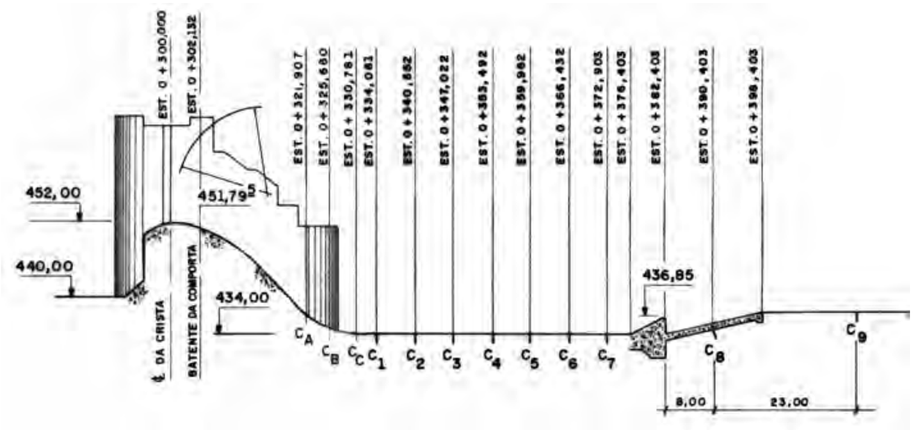

Figure 3. Pressure measurement points. Longitudinal section of the central bay.

3. NUMERICAL MODEL

The CFD program FLOW-3D, by Flow Science Inc., was used. FLOW-3D is a finite difference/volume, free surface, transient flow modeling system that was developed to solve the Navier-Stokes equations in three spatial dimensions. The finite difference equations are based on an Eulerian mesh of non-uniform hexahedral (brick shaped) control volumes (to form mesh-blocks) using the Fractional Area/Volume (FAVOR) method. Free surfaces and material interfaces are defined by a fractional volume-of-fluid (VOF) function (Hirt & Nichols, 1981).

In hydraulic engineering most important flows are turbulent. One of the methods used to obtain the approximate solutions of the turbulent flow equations was first proposed by Osborne Reynolds (Rodi, 1993). This statistical approach of the equations that represent the turbulent flow, results in the Reynolds-averaged Navier-Stokes equations, which application implies in the use of the turbulence models, for the representation of the new term that appears on the momentum equation.

The numerical modeling of the flow inside of the stilling basin is much complex due to the high intensity of the turbulence and the recirculation that is associated to the hydraulic jump. To represent these characteristics of the flow, it was used the k-e turbulence model. A great amount of turbulence models, for application on hydraulic engineering, is found in literature. The aeration effects in the hydraulic jump were considered in the numerical simulation.

An improved k-e turbulence model, based on Renormalization-Group (RNG) methods (Yakhot & Orszag, 1986), was used. This approach applies statistical methods for a derivation of the averaged equations for turbulence quantities, such as turbulent kinetic energy and its dissipation rate. The RNG-based models rely less on empirical constants while setting a framework for the derivation of a range of models at different scales.

The RNG model uses equations similar to the equations for the k-ε model. However, equation constants that are found empirically in the standard k-ε model are derived explicitly in the RNG model.

Generally, the RNG model has wider applicability than the standard k-ε model. In particular, the RNG model is known to describe more accurately low intensity turbulence flows and flows having strong shear regions.

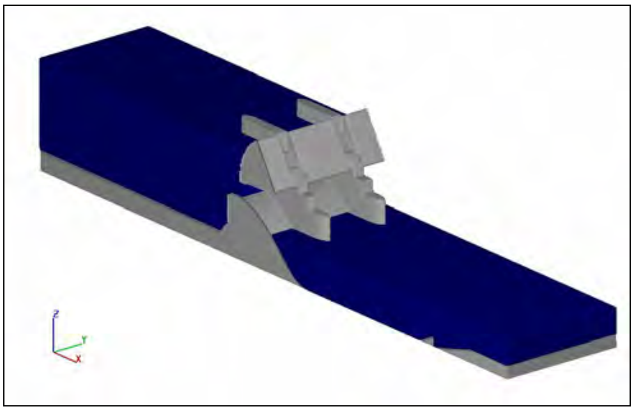

Figure 4. Model domain for the stilling basin

The boundary conditions were imposed for pressure and velocity values. On the free surface, the pressure is zero, so that every flow pressure value has relative value. On the solid surfaces, the no-slip conditions were applied. At the inlet and outlet of the domain, hydrostatic pressure distribution was assumed, and depths measured in the physical model were specified. The boundary conditions for the values associated with the k-e model; the turbulent kinetic energy, k, the turbulent energy dissipation, e, also were imposed to the numerical model. At the inlet was assumed production and dissipation nulls. The longitudinal gradients of the other variables were assumed to be zero at the downstream end. Close to the solid surface, there is the need of some special considerations due to the presence of the boundary layer. The boundary values for k and e were defined according with a law of the wall.

4. NUMERICAL RESULTS

The physical model configurations were numerically simulated until the hydraulic jump stabilization, for a given flow rate. During the simulations, the flow starts from rest and is settled by the water level difference between the upstream and the downstream. There is an initial time gap, for which the hydraulic jump, still is not settled and the characteristic flow parameters present a great time fluctuation. When the jump becomes stable, these values have a small fluctuation around an average value.

Figure 5 presents the velocity contours (horizontal component), for Q = 4000 m3/s, obtained in the 3D numerical simulation.

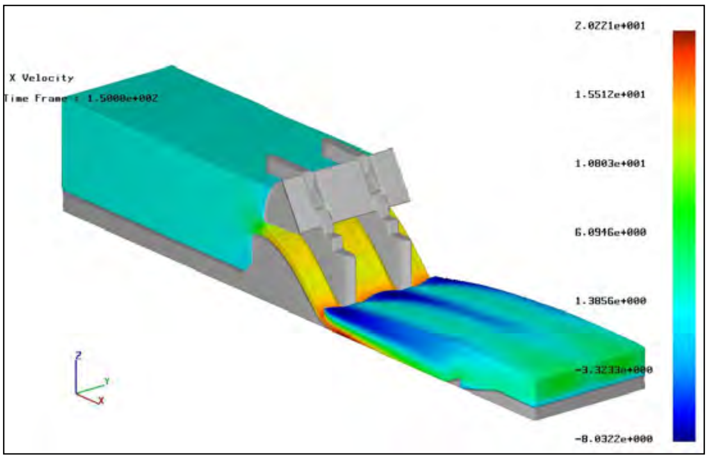

Figure 5. Horizontal component of the velocity – Q = 4000 m^3/s

Figure 6 presents the rate of the average turbulent energy dissipation along the stilling basin. It is possible to observe that the dissipation reaches its maximum value in the vicinity of the beginning of the jump roller, decreasing along the basin.

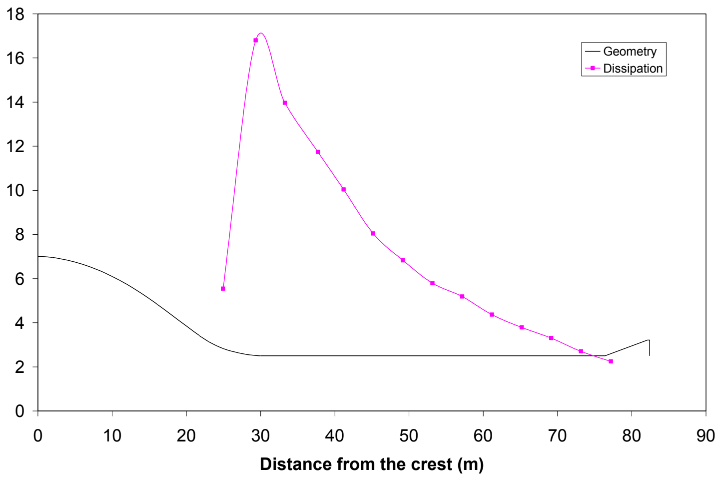

Figure 6. Turbulent kinetic energy dissipation rate - Q = 6000 m3/s

It is possible to observe that the Figure 6, with the results obtained by Marques et al. (1998), presented in Figure 7 that represents the energy dissipation along the jump; through the values of the standard deviation of the acquired samples of the instantaneous pressure, assuming that the energy dissipation is only due to the jump while there is some pressure variations, caused by its occurrence.

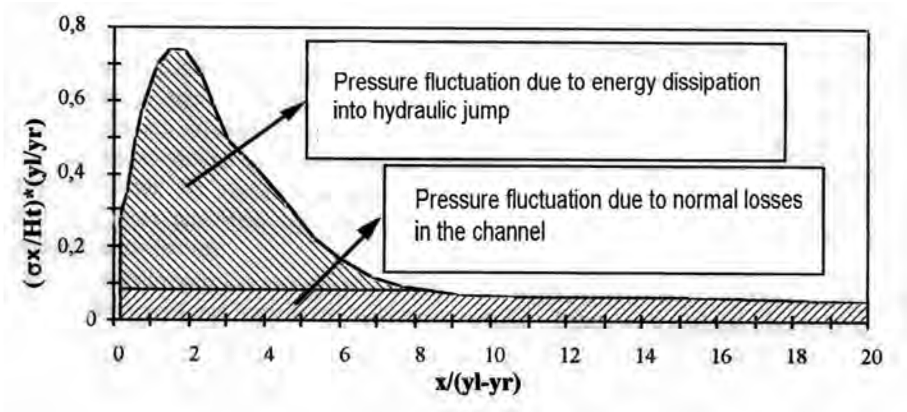

Figure 7**.** Schematic representation of the head losses in a hydraulic jump (Marques et al, 1998)

5. EXPERIMENTAL AND NUMERICAL COMPARISONS

For validating the results obtained with the numerical simulation proposed in the present paper; they were compared with the data recorded in the experiments with the physical model and the prototype.

The instantaneous pressures in the experiments were measured at the previously described points. Using the data obtained in these experiments, the corresponding mean pressures were calculated.

Figure 8 shows the comparison of observed and numerically modeled data for pressure, with a discharge of 4000 m3/s. The results present good agreement between the prototype, scale model and numerical simulation.

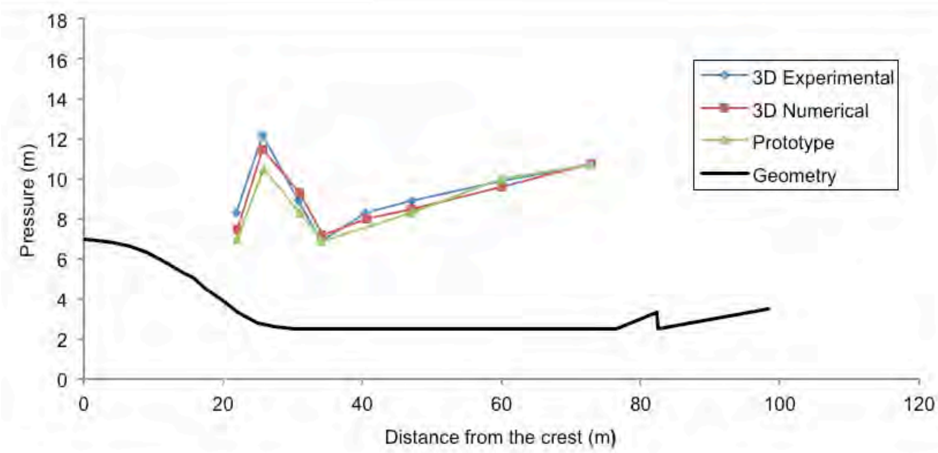

Figure 8**.** Pressure at the prototype and physical and numerical models - Q = 4000 m3/s

Figure 9 present the comparison of the water levels obtained in the numerical and physical models. The water levels comparisons between the physical and numerical models were good.

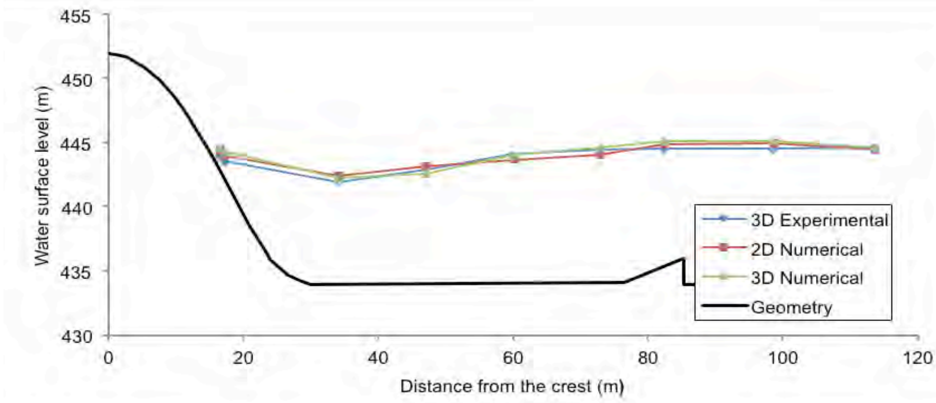

Figure 9**.** Water levels in the physical and numerical models - Q = 4000 m^3/s

6. CONCLUSIONS

The behavior of a flow inside a hydraulic jump stilling basin was described in this paper, through the CDF modeling techniques, using a physical model to validate the results.

The studies about the stilling basins are of great importance for the construction, maintenance and safety of the large hydraulic structures. The increase of the efficiency of the energy dissipation has shortened the risk of the downstream erosion of the structure that can be a danger to its stability and also contributes to the environment.

In this work was used the 1:100 scale three-dimensional physical model of the Porto Colombia Hydropower Plant, built at the Experimental Hydraulics Laboratory of Furnas S.A. (LAHE). As numerical model, was used the software Flow-3D.

Some characteristic operation conditions of the stilling basin were analyzed in the physical model. Then, were measured the water levels, the mean velocities and the instantaneous pressures, in various points of the flow. The same conditions were simulated with the numerical model, with the objective of comparing these two models.

Despite the limitations and the complexity of the geometry, the behavior of the turbulent flow inside the stilling basin was well represented by the numerical model. The model reproduced the recirculation due to the formation of the roller associated to the hydraulic jump, the small downstream recirculation just after the terminal sill, the variation of the initial position of the jump with the variation of the imposed discharge, among other aspects.

REFERENCES

Cook, C.B., Richmond, M.C., Serkowski, J.A. and Ebner, L.L. (2002). *Free-Surface Computational Fluid Dynamics Modeling of a Spillway and Tailrace: Case Study of the Dalles Project*, Hydro Vision 2002, Pacific Northwest National Laboratory, Richland, Washington.

Furnas Centrais Elétricas S.A., (1995). *UHE de Porto Colômbia: Estudos Hidráulicos em Modelo Reduzido – Modelo de Conjunto–Escala 1:100–Levantamento da Capacidade de Vazão do Vertedouro*, Internal Report, Rio de Janeiro, 28 p.

Furnas Centrais Elétricas S.A., (1996). *UHE de Porto Colômbia: Estudo de Flutuação de Pressão na Bacia de Dissipação – Campanha de Ensaios Realizados no Protótipo (in-situ),* Internal Report, Rio de Janeiro, 28 p.

Higgs, J.A. (1996). *Type II and III stilling basin modifications computational fluid dynamic model study*, Water Resources Research Laboratory, Bureau of Reclamation, The Stilling Basin Abrasion Damage Prevention Enterprise Project.

Hirt, C.W. and Nichols, B.D., *Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*, J. Comp. Phys., Vol. 39, 1981, pp.201-225.

Marques, M.G., Ollermann, G., Weiller, C. and Endres, L.A.M., (1998). *Perda de carga no interior de um ressalto hidráulico a jusante do vertedouro*, Congresso Latino Americano de Hidráulica, 1998, Oaxaca, México.

Peterka, A.J., (1984). *Hydraulic design of stilling basin and energy dissipators.* 8. ed., Denver, Colorado: United States Government Printing Office, 222 p.

Preslan, W. and Wilhelms, S. (2001). *Dalles – Metric Feseability Study*, ERDC – Engineer Research and Development Center, USACE – U.S. Army Corps of Engineers, Vicksburg, Mississippi.

Rodi, W. (1993). *Turbulence Models and their Application in Hydraulics: A State-of-the-Art Review*, 3. ed., A.A. Balkema, Rotterdam, 103p.

Yakhot V. and Orszag S.A., (1986). *Renormalization Group Analysis of Turbulence. I. Basic Theory*, J. Scientific Computing, 1, 1986, pp. 1-51.