A NUMERICAL AND EXPERIMENTAL STUDY OF HYDRAULIC JUMP STILLING BASIN

José Carlos C. Amorim[1], Renata C. Rodrigues[2] and Marcelo G. Marques[3]

ABSTRACT

This paper presents a study about hydraulic jump stilling basins, using a physical model of the Porto Colombia Power Plant basin and a computational fluid dynamics (CFD) model. The Furnas Centrais Elétricas S.A. constructed and operates the Porto Colombia Hydroelectric Power Plant. These studies had two main goals. The first goal is to determine if CFD is an effective tool to evaluate the potential for abrasion in stilling basins. The second goal is to determine if the additional tools of CFD augment the physical stilling basin investigation, such as plots of flow fields, will improve understanding of the transient nature and optimize the recommended modifications. To simulate the highly transient and turbulent flow conditions in the stilling basin, a free-surface CFD numerical model has been applied. This model is based on the volume-of-fluid (VOF) method, and is capable of simulating sudden discontinuities in the free surface, including wave breakup. The model solves the non-hydrostatic Reynolds-averaged Navier-Stokes (RANS) equations over variable-sized hexahedral cells. To represent the characteristics of the turbulence, it was used a k-ε turbulence model. To validate the ability of the numerical model to simulate flows downstream of the spillway, the model was validated against data from a physical model at scale of 1:100.

1. INTRODUCTION

Traditionally, the studies of the performance of the hydraulic jump stilling basins are carried on only 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.

With the recent 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.

---

[1] Associate Professor, Military Institute of Engineering – IME, 22290-270 Rio de Janeiro – RJ, Brazil (jcamorim@ime.eb.br)

[2] Engineer, Furnas Centrais Elétricas S.A., 22723-400 Rio de Janeiro – RJ, Brazil, (renatacr@furnas.com.br)

[3] Associate Professor, Hydraulic Research Institute – IPH/UFRGS, 91540-000 Porto Alegre – RS, Brazil (mmarques@iph.ufrgs.br)

Numerical simulations of hydraulic jumps using the integral approach were proposed by Narayanan (1975) and McCorquodale and Khalifa (1983). In hydraulic jumps, the turbulence production and diffusion processes are very complicated in nature; numerical simulations using the Reynolds-averaged Navier-Stokes equations (RANS) allow simulating details of the flow inside the jump. In 1991 Long et al. (1991) made a numerical study of a submerged hydraulic jump, using a control volume method and a k-ε turbulence model. Chippada et al. (1994) developed a model for hydraulic jumps in open channels using the finite element method and also the standard k-ε turbulence model, but the results obtained were not compared whit any experimental results. Liu and Drewes (1994) investigated the characteristics of the turbulence on free and forced jumps using a k-ε turbulence model.

The present work presents a numerical and experimental study of the turbulent flow in a hydraulic jump stilling basin. A physical model and a numerical model, based on free-surface CFD (Computational Fluid Dynamics) model, are used in the development.

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 was, the Porto Colombia Hydro Station that is a three-dimensional 1:100 scale model, built at the Experimental Hydraulic Laboratory of Furnas S.A. (LAHE) in 1992.

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.

In the bibliographic review that was realized, the only related work, that was found, was the developed by The Water Resources Research Laboratory of The Bureau of Reclamation, of the United States (Higgs, 1996). The paper, “Type II and Type III Stilling Basin Modifications Computational Fluid Dynamic Model Study” was published, as a result of this study. 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 (Flow-3D) were compared, to study the return flows that may carry abrasive material to the inner stilling basin.

2. PHYSICAL MODEL

2.1 The Porto Colombia Dam Physical Model

This 1:100 scale model occupies a surface of 120 m^2 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. The Creager profile spillway 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.

2.2 Test Program

A schedule was elaborated, for the development of the tests; it determines the conditions for the realization of the tests with the physical model, to be simulated after, with the numerical model. The determination of the conditions for the experiments was based in normal conditions of operation of the plant, focusing the released discharge and the forebay and tailwater elevation. The instantaneous pressures, the mean velocities and the water levels, were measured for all situations tested.

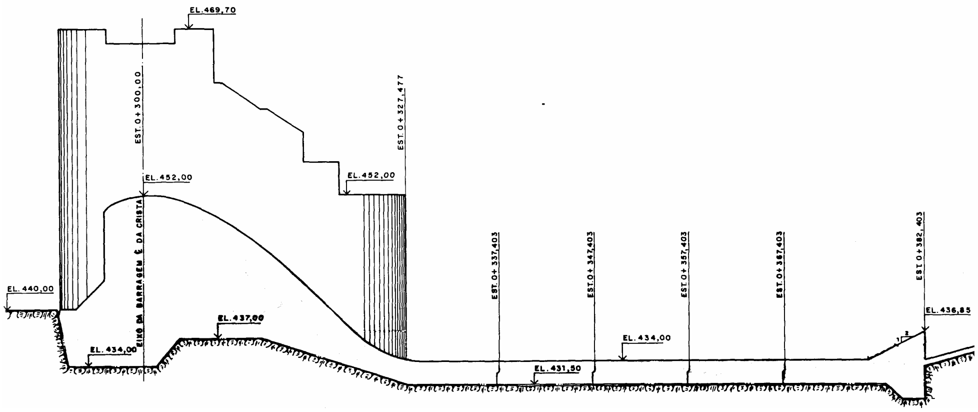


Figure 1 Longitudinal section of the Porto Colombia's spillway and basin.

2.3 Measurement Positions

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 2.

The mean velocities at the points C1, C3, C5, C7, C8, and C9 were measured. The two last points are located outside the basin. Velocities were measured at 1 cm off the bottom. Micro current meters were used for the velocity measurements. The water levels at the longitudinal section of the basin were measured, corresponding to central bay axis.

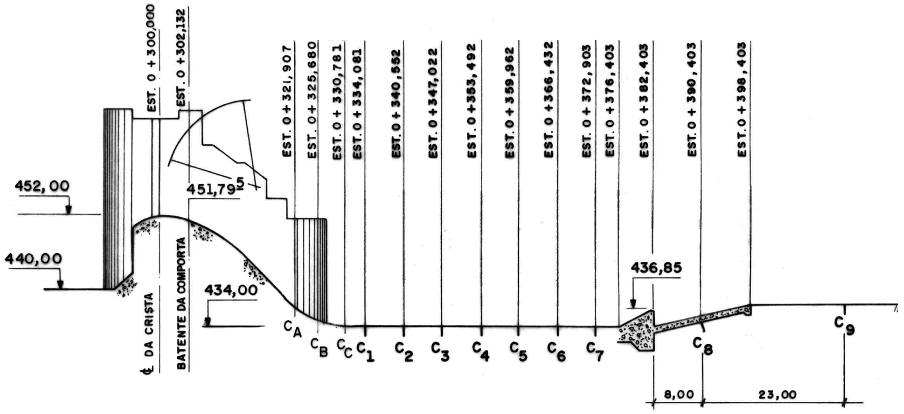


Figure 2 Pressure measurement points. Longitudinal section of the central bay.

2.4 Instantaneous Pressure at the Physical Model

Figures 3 to 5 present the results obtained in the simultaneous pressure data acquisition in the time domain, at the eight transducers for the discharges: 1000, 4000 and 8000 m^3/s.

For better visualization of the results, only 30 seconds of the 15 minutes data are being presented. Observe in Figures 3 to 5, especially for the higher discharges, the process of generation and amplification of the turbulence. The pressures in the time domain, during the 30 initial seconds of the acquisition, are represented in these figures. The average line is represented in pink; it corresponds to the pressure mean value that was obtained during the total 15 minutes of the acquisition. Also, one may observe a gradual increase of the energy due to turbulence at the basin, because of the higher pressure fluctuation, with the increase of the flow. These figures also show, by the simultaneous acquisition at the pressure transducers, the formation process of the pressure fluctuation peaks, which are carried, downstream and attenuated at the end of the basin. In Figures 3 to 5, the instantaneous pressure data reproduce the downstream displacement of the jump, with the flow increase.

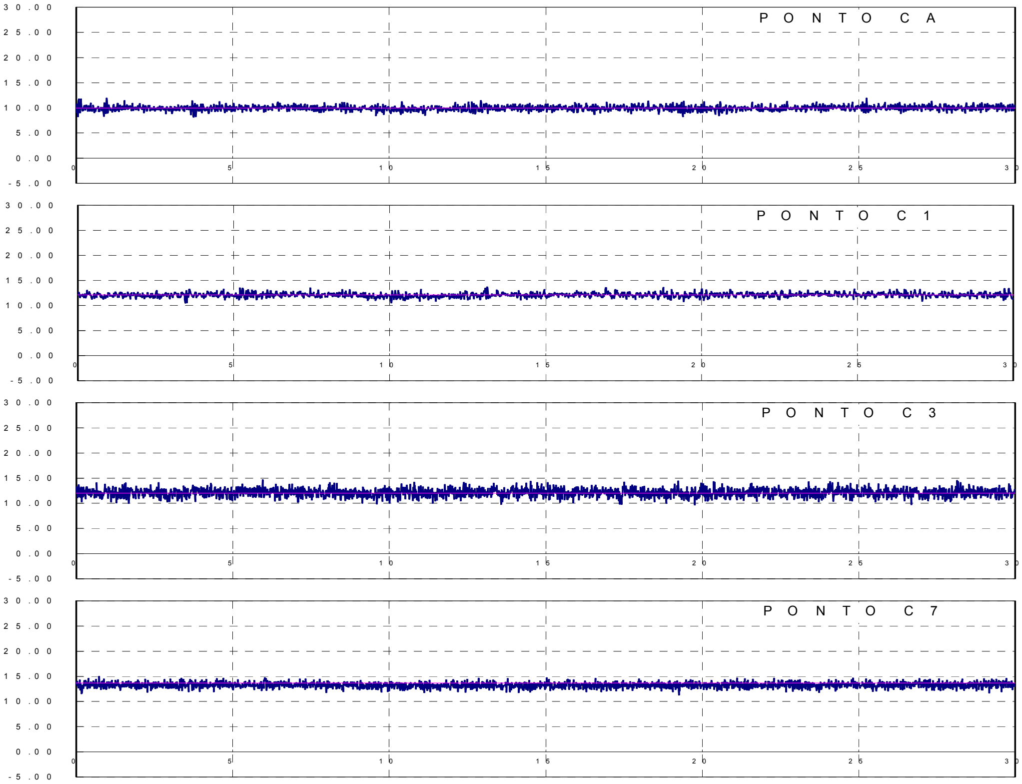


Figure 3 Instantaneous pressures - Q = 1000 m³/s.

3. NUMERICAL MODELING

The commercial software Flow-3D, developed by Flow Science, was used for the numerical modeling of the flow. The Flow-3D uses the finite-volume method to solve the Reynolds-averaged Navier-Stokes (RANS) equations over the computational domain. For each cell, values of the state variables were solved at discrete times using a staggered grid technique. Tracking of the free-surface is performed using de Volume-of-Fluid (VOF) method, which produces a surface that is free of the "stair-stepping" effect normally associated with Cartesian hexahedral grids.

3.1 Turbulence Model

In hydraulic engineering most important flows are turbulent. Despite the recent progress of the computational resources, the turbulent flows still cannot be precisely numerically treated. One of the methods used to obtain the approximate solutions of the turbulent flow equations was first proposed by Osborne Reynolds (apud 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.

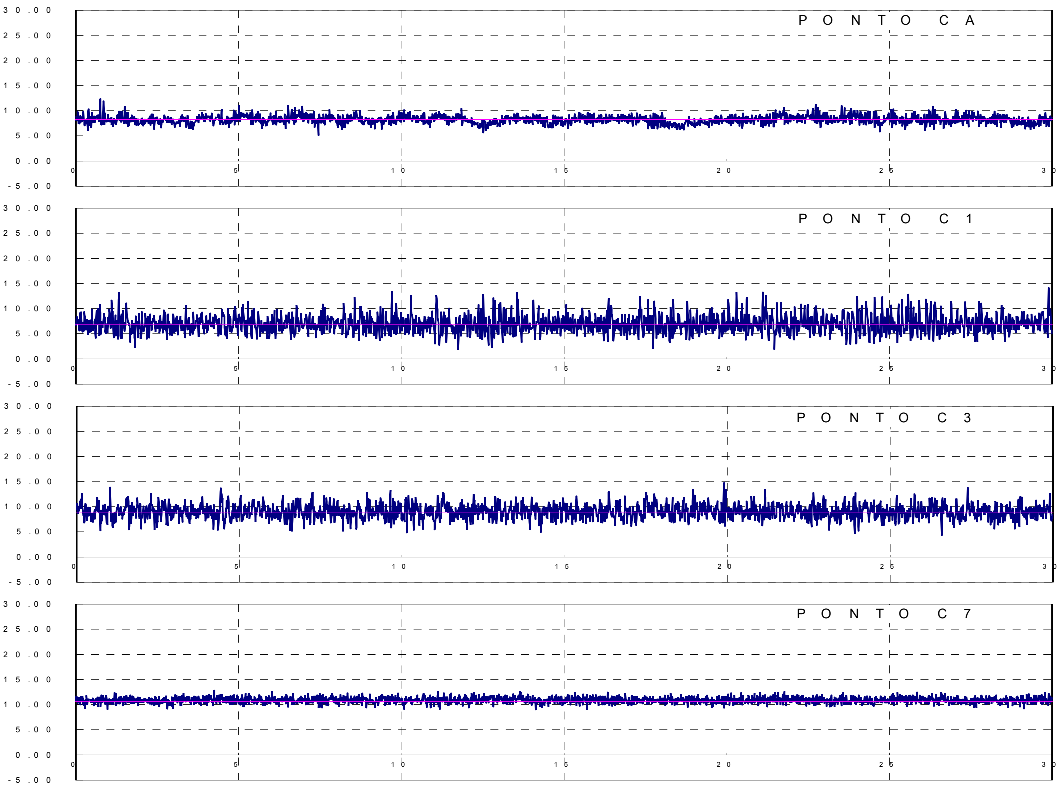
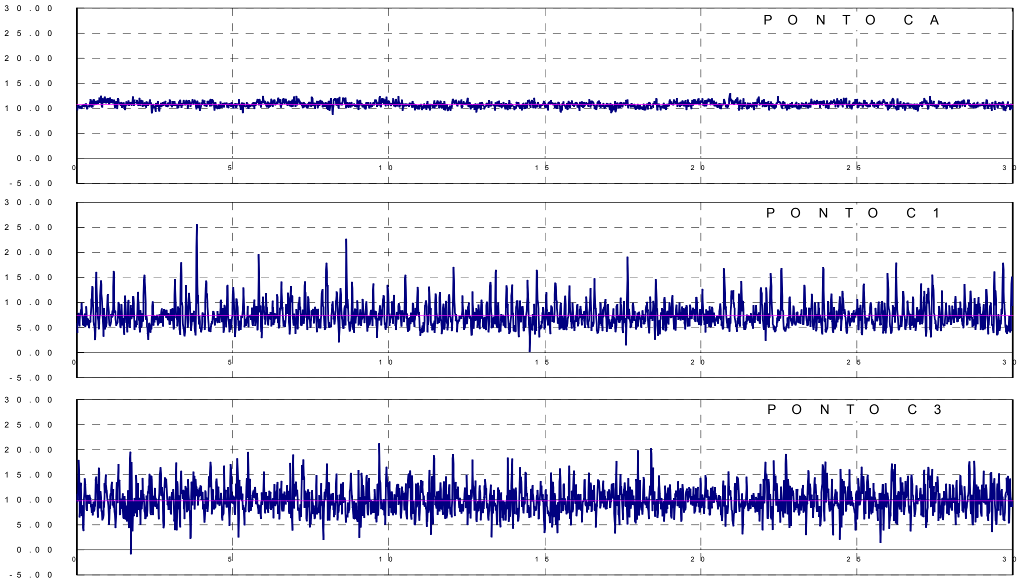


Figure 4 Instantaneous pressures - Q = 4000 m³/s.



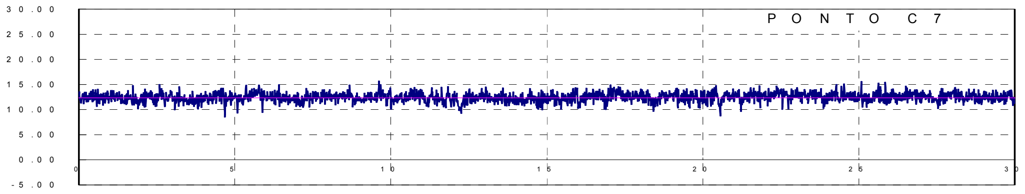


Figure 5 Instantaneous pressures - Q = 8000 m³/s.

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-ε turbulence model. A great amount of turbulence models, for application on hydraulic engineering, is found in literature. Rodi (1993) presents a detailed discussion of the applicability of various turbulence models.

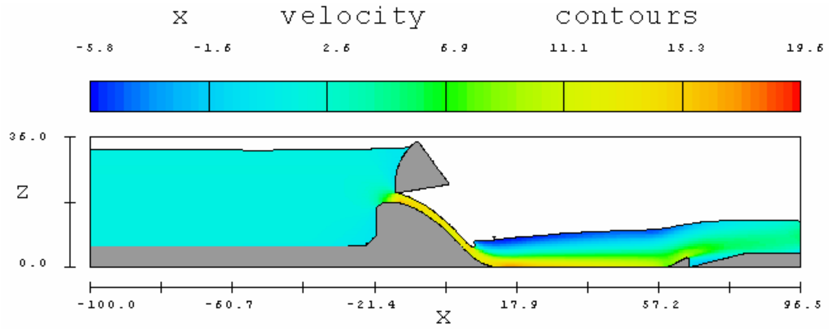
3.2 Boundary Conditions

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-ε model; the turbulent kinetic energy, k, the turbulent energy dissipation,ε, 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 ε were defined according with a law of the wall.

3.3 Numerical Simulation

The tested physical model configurations were simulated during 150 s, time gap that showed to be enough for the hydraulic jump stabilization. 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 6 shows the velocity contours (horizontal and vertical components), for Q = 4000 m^3/s.



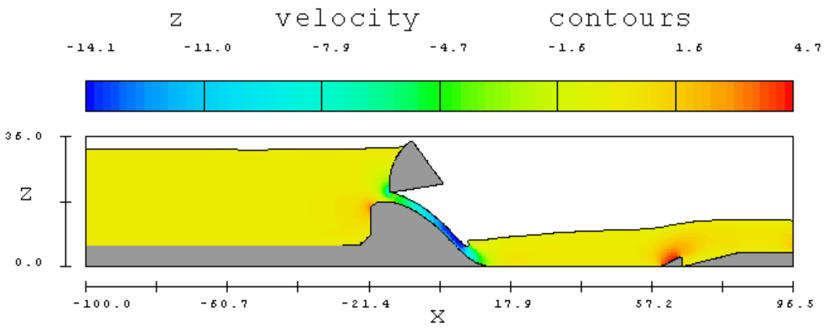


Figure 6 Horizontal and vertical components of the velocity - Q = 4000 m^3/s.

Figure 7 shows the mean velocity contour for Q = 8000m^3/s.

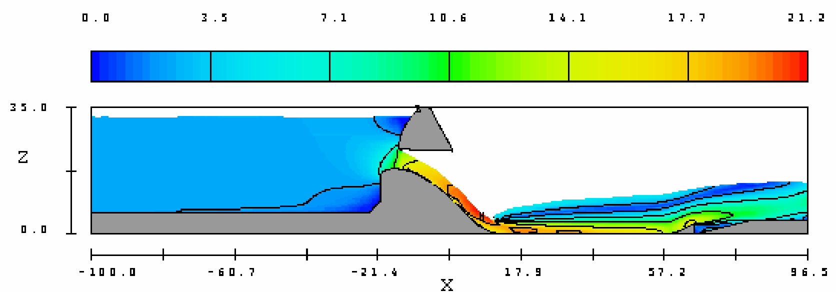


Figure 7 Mean velocity contour - Q = 8000 m^3/s.

Figure 8 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.

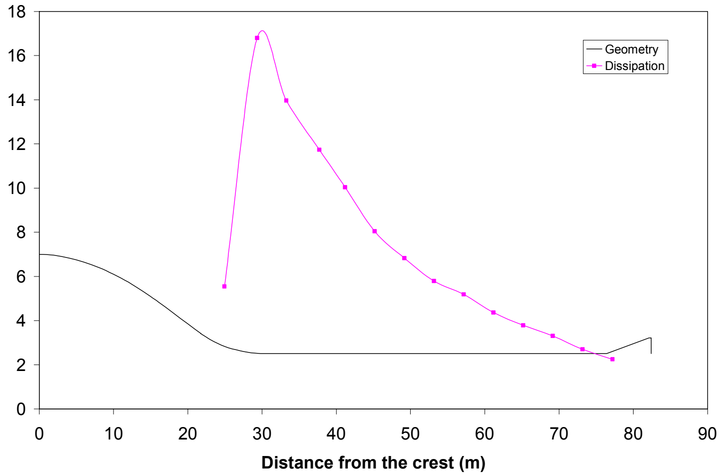


Figure 8 Turbulent kinetic energy dissipation rate - Q = 6000 m^3/s.

It is possible to observe that the Figure 8, with the results obtained by Marques et al. (1998), presented in Figure 9 that represents the energy dissipation along the stilling basin; 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.

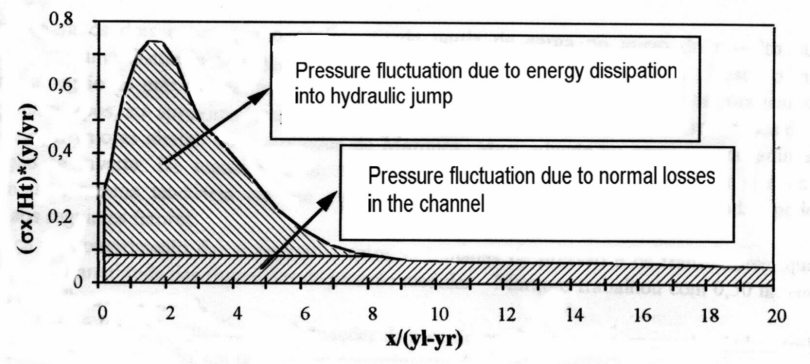


Figure 9 Schematic representation of the head losses along the jump. (Marques et al., 1998).

4. NUMERICAL AND EXPERIMENTAL RESULTS AND COMPARISONS

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

4.1 Average Pressure

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. To assure that the average values of the numerical model, correspond to the occurrence of stable jump, were used values obtained between t = 80 and 150s.

Figure 10 shows the comparison of observed and numerically modeled data for pressure, with a discharge of 8000 m^3/s. The results are in general agreement at most location, however differences do exist.

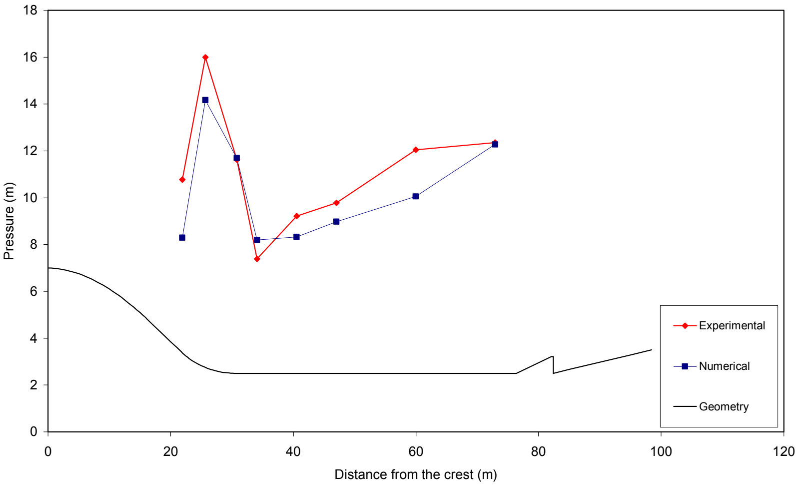


Figure 10 Pressure at the numerical and physical models - Q = 8000 m^3/s.

Figure 11 presents the measured pressure data obtained, in-loco, at the Porto Colombia Hydro Station in comparison with the numerical and 1:100 scale physical models, for a flow of 4000 m^3/s.

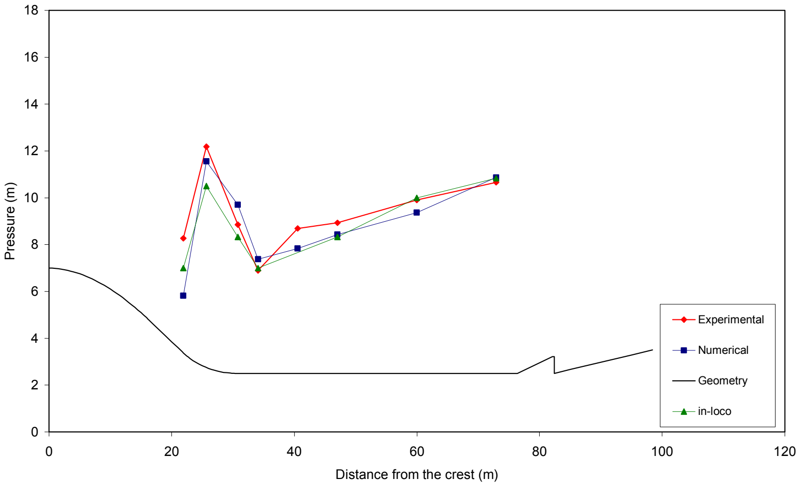


Figure 11 Pressure at the numerical and physical models - Q = 4000 m^3/s.

It is convenient to observe that the air entrainment process, caused by the formation of the hydraulic jump, turns the process in a two-phase flow; and that the effects of this process were not considered in the numerical model. Moreover, the numerical model used in this paper is two-dimensional and it does not consider the possible three-dimensional effects of the flow inside the stilling basin.

5. 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 Hydro Station, built at the Experimental Hydraulic 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

Chippada, S., Ramaswamy, B. and Wheeler, M. F. (1994). “Numerical simulation of hydraulic jump”, International Journal for Numerical Methods in Engineering, Vol. 37, No. 8, pp. 1381-1397.

Endres, L. A. M. (1990). Contribuição ao Desenvolvimento de um Sistema para Aquisição e Tratamento de Dados de Pressões Instantâneas em Laboratório, M.Sc. Thesis – UFRGS – IPH, 104 p.

Fiorotto, V. and Rinaldo, A. (1992). "Turbulent pressure fluctuations under hydraulic jumps", Journal of Hydraulic Research, Vol. 30, No. 4, pp. 499-520.

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-loco*), Internal Report, Rio de Janeiro, 28 p.

Furnas Centrais Elétricas S.A. (1997). UHE de Porto Colômbia: Estudo de Flutuação de Pressão na Bacia de Dissipação – Campanhas de Ensaios Realizadas no Modelo de Conjunto – Escala 1:100 – Pressões Médias, Internal Report, Rio de Janeiro, 3 p.

Higgs, J. A. (1996). Type II and III Stilling Basin Modifications Computational Fluid Dynamic Model Study, Water Resources Research Laboratory, Bureau of Reclamation - USBR, The Stilling Basin Abrasion Damage Prevention Enterprise Project.

Khader, M. H. A. and Elango, K. (1973). "Turbulent Pressure Field Beneath a Hydraulic Jump", Journal of Hydraulic Research, Vol. 15, No. 4, pp. 469-489.

Liu, Q. and Drewes, U. (1994). "Turbulence Characteristics in Free and Forced Hydraulic Jumps", Journal of Hydraulic Research, Vol. 32, No. 6, pp. 877-898.

Long, D., Steffler, P. M. and Rajaratnam, N. (1991). "A Numerical Study of Submerged Hydraulic Jumps", Journal of Hydraulic Research, Vol. 29, No. 3, pp. 293-308.

Lopardo, R. A. (1987). "Notas sobre Fluctuaciones Macroturbulentas de Presión, Medición, Análisis y Aplicación al Resalto Hidráulico", Revista Latino Americana de Hidraulica, No. 2, pp. 109-154.

Marques, M. G., Drapeau, J. and Verrete, J. L. (1997). "Flutuação de pressão em um ressalto hidráulico", Revista Brasileira de Engenharia, Vol. 2, pp. 45-52.

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", In: Congresso Latino Americano de Hidráulica, Oaxaca, México.

McCorquodale, J. A. and Khalifa, A. (1983). "Internal Flow in Hydraulic Jumps", Journal of Hydraulic Engineering, Vol. 109, No. 5, pp. 684-701.

Narayanan, R. (1975). "Wall Jet Analogy to Hydraulic Jump", Journal of the Hydraulics Division, ASCE, Vol. 101, No. HY3, pp. 347-360.

Rouse, H., Tien To Siao and Nagaratnam, S. (1959). "Turbulence Characteristics of the Hydraulic Jump", Transactions of the ASCE, Vol. 124, pp. 926-950.

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

Toso, J. W. and Bowers, C. E. (1988). "Extreme Pressures in Hydraulic Jump Stilling Basins", Journal of Hydraulic Engineering, Vol. 114, No. 8, pp. 829-843.