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

COMPLEMENTARY SPILLWAY OF SALAMONDE DAM. PHYSICAL AND 3D NUMERICAL MODELLING

MIGUEL R. SILVA(1) & LÚCIA T. COUTO(2) & ANTÓNIO N. PINHEIRO(3)

(1) *Civil Engineer, Aqualogus, Engenharia e Ambiente, Lda., Lisbon, Portugal*
miguel.r.silva7@gmail.com

(2) *Research Officer, LNEC, National Laboratory for Civil Engineering, Lisbon, Portugal*
lcouto@lnec.pt

(3) *Full Professor, IST, University of Lisbon, Lisbon, Portugal*
antonio.pinheiro@tecnico.ulisboa.pt

Abstract

Throughout the planning and design of hydraulic structures, engineers and researchers are increasingly integrating computational fluid dynamics (CFD) into the process. Despite reports of success in the past, there is still no comprehensive assessment that assigns ability to CFD models to simulate a wide range of different spillways configurations.

The complementary spillway of Salamonde dam, located in the north of Portugal, is controlled by an ogee crest and two radial gates, followed by a free surface flow tunnel, with a rather complex geometry, and a terminal ski jump which directs the jet into the river bed. The present paper analyses the ability of a CFD model (FLOW-3D) to simulate the flows along this spillway.

The spillway was primarily tested and developed in a physical model built in the National Laboratory for Civil Engineering (LNEC), where discharges and flow depths were measured in ten defined cross-sections for four different gate openings conditions. These results were used to calibrate the numerical model and to analyze the differences between physical and numerical models results.

It is shown that there is an accurate agreement between physical and numerical model discharges. Concerning the flow depths, the FLOW-3D represents reasonably well the flow behavior, but slightly underestimates the flow depth in some points of the cross-sections. A sensitivity analysis for the conditions and parameters of the numerical model (e.g., 1st vs 2nd order momentum advection, turbulent mixing length TLEN, mesh size) was carried out. According to FLOW-3D results, for a certain reservoir level, spillway discharge and flow depth are highly dependent on mesh size.

Conclusions about the most adequate FLOW-3D options to adopt are presented. The calibrated model was used to simulate the spillway design discharge and to assess the hydraulic behavior of the outlet structure and of the jet impingement characteristics. The pressure distribution in specific cross-sections was also assessed.

Keywords: Spillway; CFD models; Turbulence; Physical model; FLOW-3D.

1. Introduction

Throughout the design and planning of hydraulic structures as spillways, some technical decisions are often supported by assessing the flow behavior in physical models. However, engineers and researchers are increasingly integrating computational fluid dynamics (CFD) into the process. Despite reports of success in the past, there is still no comprehensive assessment that assigns ability to CFD models to simulate a wide range of different spillways configurations. In this study, the applicability of a commercial CFD model (FLOW-3D) to simulate the flow into a complex geometry spillway was reviewed.

To solve the governing equations of fluid flow, FLOW-3D solves a modification of the commonly used Reynolds Average Navier-Stokes (RANS) equations. Additionally, the software includes algorithms to track the free surface (VOF method) and represent the geometrical details (FAVOR).

The main objective of the present study is to evaluate how accurately the FLOW-3D can represent the flow characteristics (e.g. velocity, pressure, flow depths) for a complex spillway and help engineers to make some technical decisions. A second objective is providing a sensitivity analysis for some of the main parameters/options of the model.

2. Complementary spillway of Salamonde dam

Salamonde dam, located in the north of Portugal in river Cávado, Peneda Gerês National Park, is a double-curved arch dam in concrete with a maximum height of 75 m from foundation. After safety analysis from (EDP, 2006), it was concluded that an additional discharge structure would be necessary – the complementary spillway for Salamonde dam (under construction), which is the case study assessed in this paper.

The complementary spillway of Salamonde dam, is a gated spillway, controlled by an ogee crest, followed by a tunnel with rather complex geometry, designed for free surface flow, and a ski jump structure which directs the jet into the river bed. The ogee crest is divided into two spans controlled by radial gates, with 6.50 m wide each. The design discharge for this structure is 1233 m^3/s, corresponding to a level of 270.64 m in the reservoir.

The outlet structure is a ski jump, producing a free jet with impinging region in the center of the river bed. Along the last 28 m of the outlet structure, the spillway cross-section is progressively reduced, to increase jet height, and consequently, the length of the impact area.

The Salamonde spillway was primarily tested and developed in a physical model (see Figure 1) built in the Portuguese National Laboratory for Civil Engineering (LNEC), where discharges and flow depths were measured in ten cross-sections (see Figure 4) for five different gate openings. Additionally, the pressures at some points of the bottom and side walls of the outlet structure were measured.

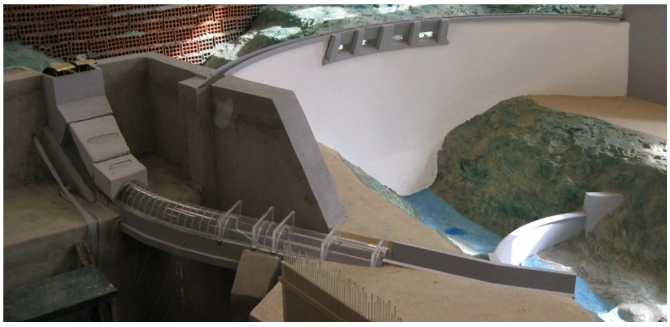

Figure 1. Salamonde spillway physical model.

3. Numerical model

3.1 General features

The numerical model used to simulate Salamonde complementary spillway flows was the FLOW-3D, which is a general purpose CFD software for modeling multi-physics flow problems, heat transfer and solidification, based on the Finite-Volume Method to solve the Reynolds Averaged Navier-Stokes (RANS) equations of the fluid motion in Cartesian coordinates. For each cell, average values for the flow parameters (pressure and velocity) are computed at discrete times using a staggered grid technique (Flow3D, 2010).

The geometrical models for spillway structure and terrain were produced using AUTOCAD 3D and AUTOCAD CIVIL 3D, respectively (Figure 2). The entire geometrical domain in (x,y,z) had the dimensions of 305 x 506 x 110 m. The mesh grid was composed by five different mesh blocks, including one nested-block, all of them with cubic cells (Figure 3). The nested-block allowed reducing the cell size at the inlet structure and inside the tunnel (in yellow at Figure 3).

The boundary conditions were specified pressure at the reservoir and upstream of the river (to represent the natural river flow), through the definition of the corresponding water levels, outflow downstream of the river and symmetry between mesh blocks.

3.2 Sensitivity analysis based on physical model discharge results

The calibration and validation of numerical models are extremely important and therefore it constitutes part of the analysis tasks in most of the CFD models. In fact, an on-going effort to carry out validation against published or experimental data remains essential. This is really important to ensure modeling correctness and to provide a high confidence level in its application.

Firstly, a sensitivity analysis of the main models/configurations in FLOW-3D based on the physical model discharge results was carried out. The model configurations under analysis were:

- Momentum advection model (1st order vs 2nd order with monotonicity preserving (MP));
- VOF method (Default vs Split Lagragian Method);
- Turbulence mixing length (TLEN) parameter;
- Cell size.

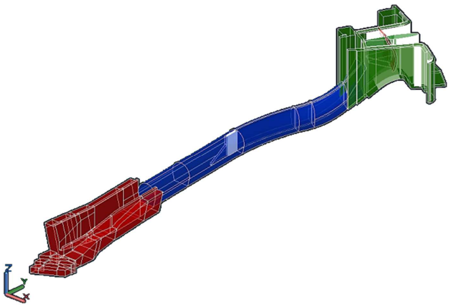

Figure 2. Salamonde spillway 3D geometrical model.

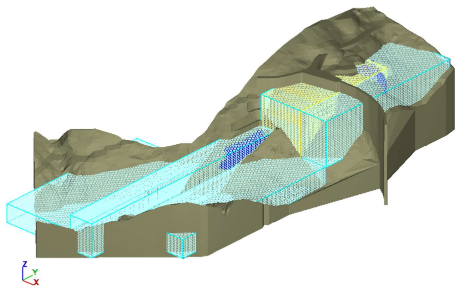

Figure 3. Numerical model domain.

In this analysis, three different discharges were considered by setting the level in the reservoir (Table 1).

Table 1. Physical model discharges for three defined reservoir levels.

RESERVOIR LEVEL (m)	$Q_{\text{PHYSICAL MODEL}}$ (m^3/s)
260.43	100.0
262.47	250.0
267.05	750.0

It is important to refer that, during the process of model configurations analysis, the simulations were performed with a 1.00 m cell size (coarsest mesh capable to solve the flow) during 200 s for sensitive analysis of momentum advection model, VOF method and TLEN parameter. Referring to mesh size sensitive analysis, 0.50 and 0.25 m cell sizes were also tested considering restart simulations of 30 s from steady state conditions. The discharges presented are calculated as an average value after steady state was attained.

Due to the large amount of data, only the more accurate results, after sensitivity analysis, will be presented for the following simulation configurations:

- 2nd order with MP momentum advection method
- Default VOF Method
- RNG turbulence model
- TLEN = 0.4 m
- Cell size = 0.25 m

The numerical results are presented in Table 2. Additionally, the differences to physical model results are shown.

Table 2. Numerical model discharges.

RESERVOIR LEVEL (m)	$Q_{\text{PHYSICAL MODEL}}$ (m^3/s)	Q_{CFD} (m^3/s)	Diff. (%)
260.43	100.0	90.1	-9.9
262.47	250.0	235.2	-5.9
267.05	750.0	729.6	-2.7

Up to more accurate configurations presented above, several combinations of parameters were tested with FLOW-3D, resulting in 14 as a total number of simulations. Throughout this process, some main conclusions were obtained:

- 2nd order momentum advection with monotonicity preserving showed more accurate results without much more computational effort.
- Split Lagrangian VOF had almost no influence on the discharge results and needed significantly more computational effort.
- The transition from a laminar regime model to a turbulence one had, naturally, influence on the discharge results. Discharges were, as expected, lower after RNG turbulence model was used.
- The turbulence mixing length parameter (TLEN) had huge importance in the simulation stability and duration; however, it did not present significant influence on discharge values.
- The obtained discharges were highly dependent on cell size.

3.3 Sensitivity analysis based on physical model flow depths measurements

In the second phase of the calibration of the numerical model, flow depths from physical and numerical model were compared. In the physical model, flow depths were measured in ten cross-sections (Figure 4) for five different gate openings. The results used to calibrate the numerical model were the ones corresponding to symmetric gates openings of 2.00 m. For these flow conditions, the physical model discharge was 267.0 m^3/s.

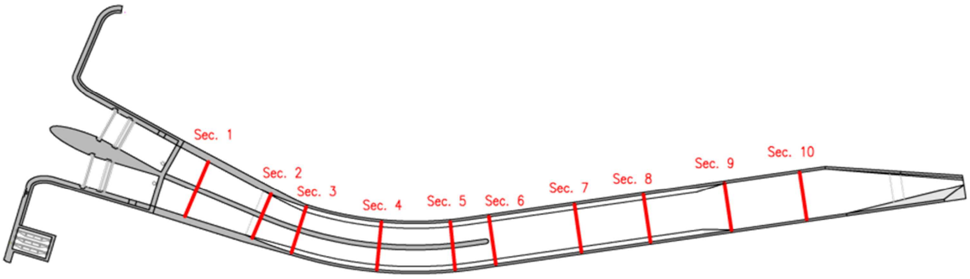

Figure 4. Physical model. Location of the ten cross-sections for measurement of flow depth (LNEC, 2012).

The model configurations under analysis were:

- Cell size;
- VOF method (Default vs Split Lagragian Method);
- Turbulence mixing length (TLEN) parameter.

The physical and computed flow depths results are compared in Figure 5. Due to the large amount of results, the ones presented are related to the cell size sensitivity analysis. It is important to refer that the presented results are related to restart simulations from steady state conditions.

The results showed that for 0.25 m cell size, FLOW-3D was able to represent accurately the flow depths although underestimates the highest values near the walls (particularly in sections 5 and 7). Clearly, the 1.00 m cell size has proved insufficient to represent accurately the flow behavior and the geometric details of the structure. One of the reasons for underestimation of flow depths by the numerical model in certain regions seems to be cell size.

In fact, cells smaller than 0.25 m should be tested to represent the flow behavior more accurately. This approach was not considered in the present research due to computational effort limitations. Actually, according to a qualitative assessment of physical model flow behavior (Figure 6 and 7), it is possible to conclude that the numerical model underestimated flow depths in flow regions with lower depths.

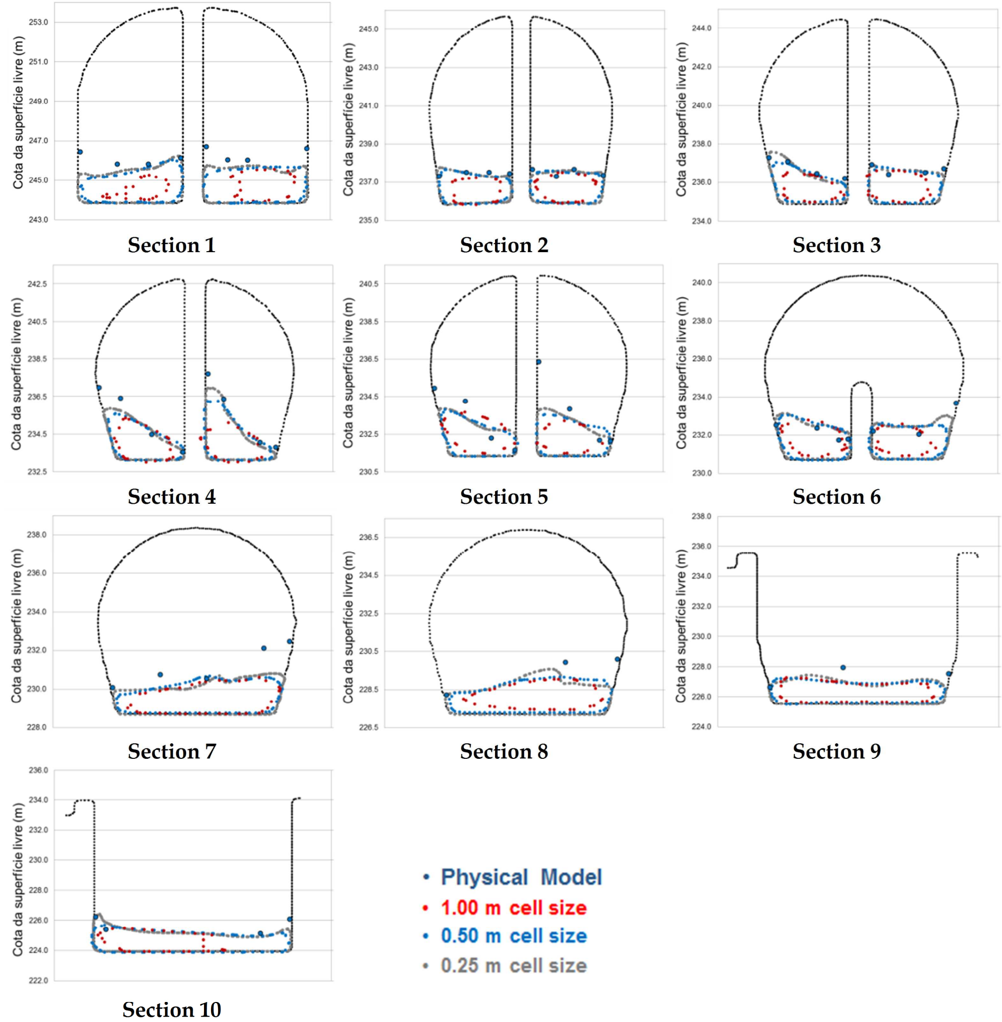

Figure 5. Flow depths for numerical and physical models. Cell size sensitivity analysis.

The numerical flow rate results are presented in Table 3 as a function of cell size.

Table 3. Numerical model discharges as a function of the cell size.

$Q_{Physical\ Model}$ (m^3/s)	Cell size (m): 1.00		Cell size (m): 0.50		Cell size (m): 0.25	
	Q_{CFD} (m^3/s)	**Diff.** (%)	Q_{CFD} (m^3/s)	**Diff.** (%)	Q_{CFD} (m^3/s)	**Diff.** (%)
267.0	207.8	-22.2	239.2	-10.4	259.5	-2.8

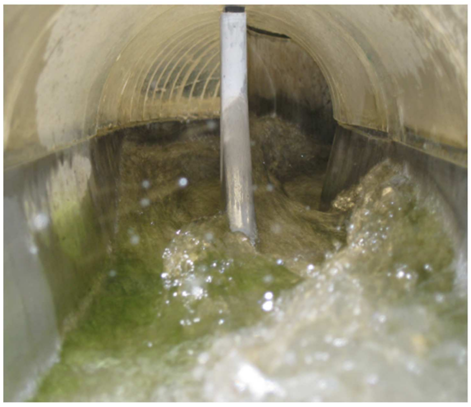

Figure 6. Q=267.0 m^3/s. Physical model.

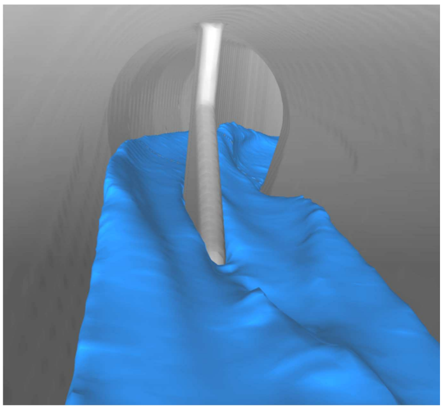

Figure 7. Q=267.0 m^3/s. Numerical simulation.

The results presented in Table 3 show an increased accuracy for smaller mesh size simulations.

Referring to the VOF method sensitivity analysis, the Split Lagragian VOF method was not able to reproduce the highest flow depths near the walls and the computing time was significantly increased. Additionally, the flow rate results were almost independent of the VOF method. Therefore, the Default VOF method was considered for further simulations.

Regarding TLEN parameter sensitivity analysis, this had neither influence in flow depths nor flow rate results. The dynamically computed option of TLEN parameter (default option) did not appear to be a reasonable choice since it drastically increased the computing time.

4. Computational Results for Design Discharge Conditions

4.1 General remarks and results

After having calibrated and validated the numerical model, the design discharge condition was simulated and the corresponding results were assessed. The FLOW-3D main configurations were:

- 2nd order monotonicity preserving momentum advection;
- Default VOF (One fluid, free surface);
- RNG turbulence model;
- TLEN adjusted in real time (between 0.4 and 0.1 m);
- Cell size 0.50 m and 0.25 m for Restart simulation.

The simulations and real computing time were:

1st simulation: cell size 0.50 m for 60 s corresponding to a real simulation time of 21:27 h;

2nd simulation (Restart simulation): cell size 0.25 m for 10 seconds corresponding to a real simulation time of 17:05 h.

The computed discharges calculated in the first simulation are presented in Figure 8. After steady state was attained, the average computed discharge was 1206 m^3/s, corresponding to a difference of -2.15% to the design value.

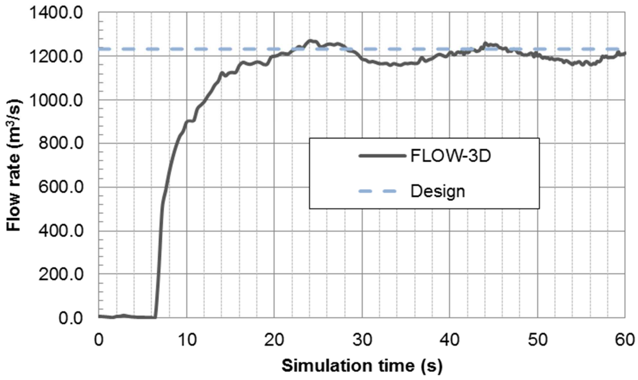

Figure 8. Calculated discharge in 1st simulation.

Due to the importance of having a global perception of the flow behavior and its main characteristics, the following topics were more carefully addressed:

- Inlet structure analysis (qualitatively);
- Flow depths;
- Velocity;
- Pressure;
- Asymmetric gates openings (qualitatively);
- Free jet;
- Water level fluctuations against the slopes of the valley.

4.2 Inlet structure analysis

The analysis of water flow in an inlet structure of a spillway is an important engineering problem. In fact, favorable approach flow conditions should be provided to the spillways in order to avoid reduced discharge capacity.

An analogy between the physical and numerical models for the inlet structure is presented in Figures 9 and 10.

Figure 9. Physical model inlet structure.

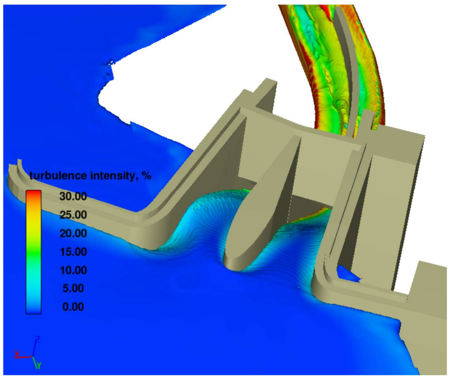

Figure 10. Numerical model inlet structure.

Figure 9 shows a slight flow separation along the pier and the guide walls. This flow pattern is also represented by the numerical model (Figure 10).

4.3 Flow depths

The quantitative flow depths analysis was carried out by comparing the numerical model results with the ones obtained in the design phase. It is important to notice that in this phase, the mean flow depths at specific cross-sections were calculated with a simple 1D approach, which considers the hydrostatic pressure distribution along the whole spillway.

The average flow depths were analyzed in the ten cross-sections defined and presented in Figure 4. For Sections 1 and 2, the numerical model significantly overestimated the average fluid depth comparatively to the design values (values 40 % higher). In this region, the flow is submitted to important accelerations and the trajectories are curvilinear, fence, deviating from hydrostatic pressure distribution. It is believed that 1D model results are not accurate enough for this kind of flow conditions.

Concerning the other analyzed cross-sections, the numerical model presented more accurate results, with an average difference of about 10 % to the design average flow depths (Silva, 2013).

4.4 Velocity

Additionally, average velocity numerical results were compared with the design ones. The post-processing software EnSight was used to render the velocity profiles in the cross-sections. Then, the local velocity mean value distribution for each cross-section was calculated using an internal function of the software. As an example, velocity profile in section 6 is presented in Figure 11.

Numerical model average velocity results showed differences of around -11 % for all ten cross-sections, comparatively to the design values (Silva, 2013). These minor differences can be partially explained by the non-uniform velocity profiles in the FLOW-3D, contrasted with the 1D model. Indeed, in the 1D model, uniform velocity profiles are considered. Therefore, it is believed that FLOW-3D can accurately represent the real velocity along the spillway.

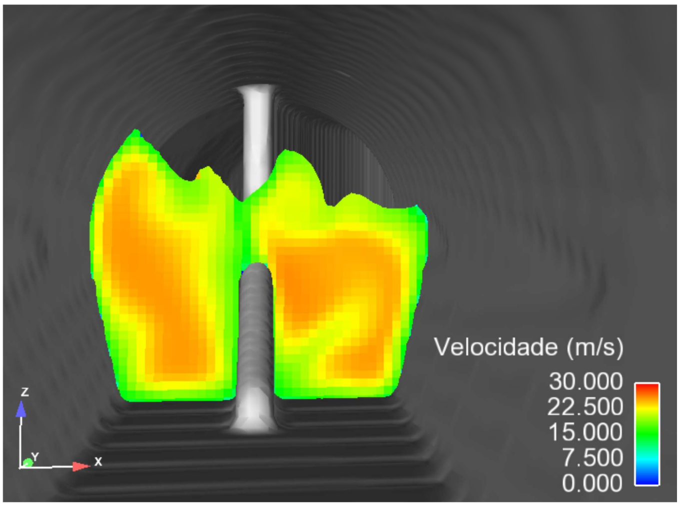

Figure 11. Velocity profile in Section 6 (rendered with EnSight®).

4.5 Pressure

One of the main objectives of the physical model was performing pressure measurements at the outlet structure. An advantage of CFD models is the ability to determine the main characteristics of the flow in any region of the computational domain. In order to assess the ability of FLOW-3D to support the structural design, a pressure analysis was developed for the outlet structure. To achieve this, pressure diagrams were rendered for each 5 m (Figure 12). Figure 12 shows, clearly, a structural conditioning section.

In the physical model, several pressure tapping points were considered in the bottom and side walls of the outlet structure. For a quantitative analysis between physical and numerical models, four different points were selected (two in the bottom and two in the right wall).

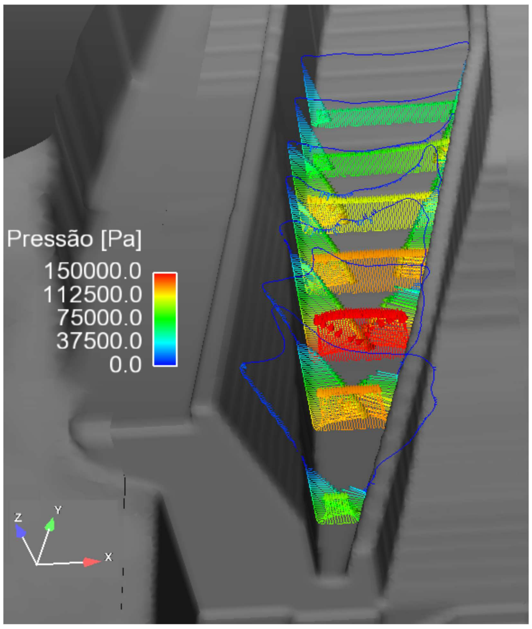

Figure 12. Pressure diagrams in different cross-sections of the outlet structure (rendered with EnSight®)

Figure 13 shows the cross-section where the highest pressure values were measured. In the physical model, the highest pressure was measured at point 2F (16.13 m water depth ≈ 158181 Pa). For this same point, the pressure computed by the numerical model presents only a difference of 0.4% (158747 Pa – Figure 13). For the remaining compared pressure tapping points (points 1, 3 and 1F), an average difference of -12.6% from numerical and physical models was achieved. It must be pointed out that hydrostatic pressure measurement is prone to errors in physical models due to slight imperfections in tap orientation relatively to the walls surfaces.

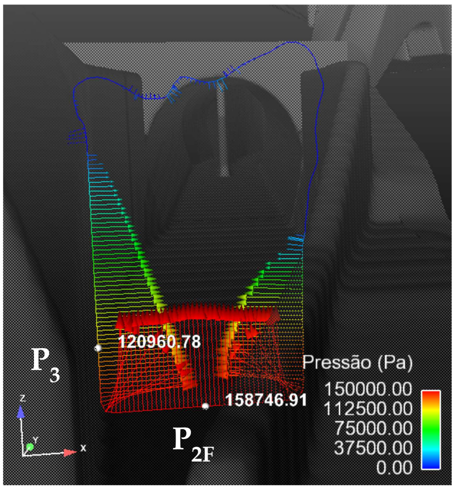

Figure 13. Pressure diagram at 10.20 m from downstream outlet structure limit.

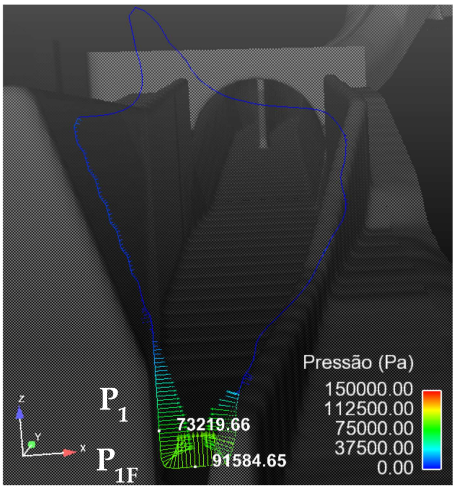

Figure 14. Pressure diagram at 2.00 m from downstream outlet structure limit.

4.6 Asymmetric gates openings

During the design phase of spillway structures, an hypothetical scenario of failure of one of the gates is often considered. Due to asymmetry of the Salamonde spillway, two different scenarios were analyzed in the physical model: right gate failure and left gate failure. These same qualitative analyzes were performed in the numerical model. The results presented in Figure 15 refer to the failure of right gate. This proved to be the worst scenario, since it showed significant flow projections against the right side wall, downstream of the septum (Silva, 2013).

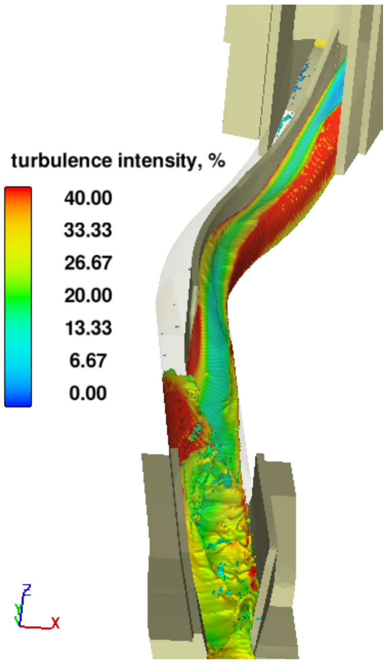

Figure 15. Flow pattern for right gate failure.

4.7 Free jet

Another important objective of the physical model experiments was to assess the jet impinging region at the river bed. In fact, in this kind of outlet structures with ski jump, engineers are concerned to direct the jet impingement to the center of the river bed to avoid significant erosions on the river banks.

Figures 16 and 17 show a comparison of jet configuration between physical and numerical models. Analyzing several physical model photos, it appears that FLOW-3D accurately represents the shapes of the jet (Silva, 2013).

Additionally, a quantitative analysis was carried out. The numerical results for minimum and maximum impingement distances of the jet were compared with the physical model measurements (Table 4).

Figure 16. Free jet. Physical model.

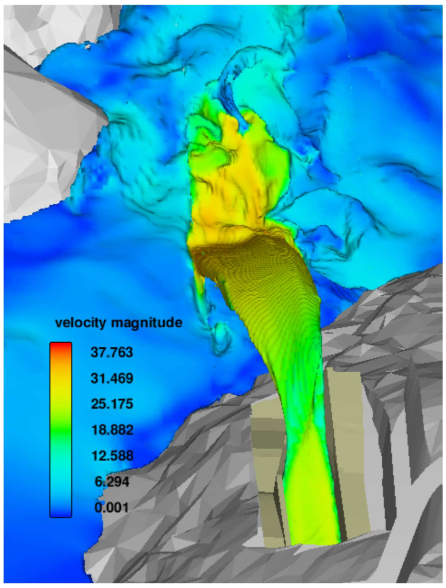

Figure 17. Free jet. Numerical model.

Table 4. Free jet impingement distance.

	Physical Model	**Numerical Model**	**Diff.** (%)
Minimum impingement distance (m)	49.0	48.5	-1.0
Maximum impingement distance (m)	91.0	68.4	-24.9

Comparing to physical model results, the numerical model accurately represented the minimum impingement distance, with a -1.0% difference. Concerning the maximum value, the difference was significantly higher, probably due to not having considered the air entrainment model available in FLOW-3D.

4.8 Water level fluctuations against the slopes of the valley

One of the major concerns with jet energy dissipation is the water level oscillations along the river slopes surrounding the plunge pool area. The numerical model can also help to decide the jet impinging region in the river bed in order to reduce erosion and water level fluctuations against the slopes of the valley.

Water level fluctuations against the slopes of the valley, after steady state was reached, are shown in Figure 18. For Salamonde complementary spillway design discharge, a 12 m elevation above the maximum river level is expected.

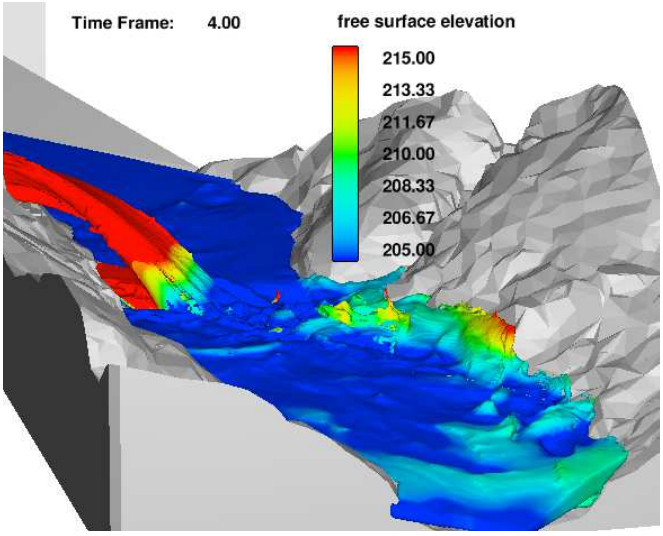

Figure 18. Water level fluctuations against the slopes of the valley.

5. Conclusions

The main conclusion from the present study is that, in general, FLOW-3D accurately simulates the flow features along a spillway with rather complex geometry, namely discharges, velocities, pressures and flow depths.

Nevertheless, there are several aspects that should be considered to obtain more accurate results. A sensitivity analysis for some main FLOW-3D configurations and parameters is highly recommended.

In the present study, the 2nd order momentum advection model with monotonicity preserving was a good choice for better accuracy, without much more computing time. Split Lagragian VOF did not appear to be useful in this kind of high velocity flow conditions. As referred in Lan (2010) and Dargahi (2010), the turbulence mixing length parameter (TLEN) has a huge importance in simulations stability and duration. The rule of thumb of 7% referred in FLOW-3D (2010) appears to be a good estimative for this parameter. However, it is recommended a sensitivity analysis to this parameter in each case study. The numerical results showed to be highly dependent of the cell size. In fact, smaller cells allow reproducing geometric details, such as complex cross-sections and radial gates, which may significantly influence the flow patterns.

As pressure diagrams and velocity profiles can be rendered with a post-processing software for any cross-section, FLOW-3D may also be very useful for structural design of hydraulic structures.

Acknowledgments

The authors acknowledge the support of Energias de Portugal (EDP) for allowing the use of the experimental data obtained by LNEC and of the design developed by Aqualogus, Consultores de Engenharia e Ambiente, Lda,. Additionally, the authors also express their gratitude to Melissa Carter and Jeff Burnham from Flow Science, Inc. support team, Francisco Garachana, Raúl Martín and Daniel Cuadra from Ensight support team and to Antonio Muralha from LNEC. Their contributions were greatly appreciated.

References

Dargahi, B., 2010. Flow characteristics of bottom outlets with moving gates. Journal of Hydraulic Research, 48(4):476–482, 2010.

EDP, 2006. Barragem de Salamonde: Controlo da segurança hidráulico operacional; revisão do estudo das cheias e análise da adequação dos órgãos de descarga. Technical report, EDP, Portugal (in Portuguese).

Flow-3D, 2010. Flow-3D User Manual, Version 10.0. Flow Science, Inc., 10 edition.

Lan, F., 2010. CFD assisted spillway and stilling basin design. In FLOW-3D Flow Simulation Contest. Flow Science, Inc.

LNEC, 2012. Modelação do escoamento no canal do descarregador de cheias complementar da barragem de Salamonde - observações em modelo reduzido. Technical report, Laboratório Nacional de Engenharia Civil (LNEC), Outubro 2012 (in Portuguese).

Silva, M., 2013. Modelação numérica 3D de escoamentos em descarregadores de cheias com escoamento em superfície livre. Descarregador complementar de Salamonde., Técnico Lisboa, Universidade de Lisboa, Julho 2013 (in Portuguese).