Second International Dam World Conference
PORTUGAL • LISBON • LNEC • April 21-24, 2015

EXPERIMENTAL AND NUMERICAL STUDY OF A CHUTE SPILLWAY.

C. Granell Ninot[†], L. Mendes[*], T.Viseu[*], J. Granell Vicent[†], A. Duque Carrero[†], J. Ortas González[‡], O. Herrero Domínguez[§], F. Río Iglesias[§] and E. Ruíz Gutiérrez[§]

[*]Laboratório Nacional de Engenharia Civil (LNEC)
Av. do Brasil, 101, 1700-066 Lisboa, Portugal
e-mail: damworld@lnec.pt, webpage: http://dw2015.lnec.pt

Keywords: Dams, Spillways, physical model, CFD numerical study.

Abstract. *This paper presents a combined approach, including CFD numerical and scale physical modelling, to address the hydraulic behavior of the spillway of a dam.*

The spillway studied is an open channel flow which crosses the right abutment of the dam, it runs in that part of the hillside and it has three separate spans controlled by Taintor Gates of 9.5 x 6.8 m. The design flow of the spillway is 1,648.5 m^3/s and the verification flow is 1,885.4 m^3/s.

Choosing and setting the spillway different forms, such as feeding, piers, gates positioning, curve at the beginning of the chute, spans convergence, height of the walls and bucket type, was performed after analyzing numerous options and through numerical methods in fluid computational hydraulics. The software used was FLOW 3D, it solves Navier Stokes equations by finite differences. The different options are now being verified on a physical scale model in the Laboratório Nacional de Engenharia Civil (LNEC).

The use of numerical methods in fluid computational hydraulics is very useful when used by the designer in combination with the physical modelling. It allows analysing different options until getting hydraulically optimal solutions which can be tested in a laboratory. This method provides undisputed advantages in the technical field as well as on the use of time and resources.

In this presentation there are a variety of approaches created by fluid computational hydraulics in relation to the spillway dam. It is analysed the hydraulic behaviour of the different options and the results of numeric model are compared with the obtained in tests carried out in the physical scale model.

[†] Jesús Granell Ing. Consultores
[‡] Eptisa
[§] Endesa Generación

1 INTRODUCTION

Traditionally, studies of complex flows in hydraulic structures have been performed through physical modeling. In those type of structures, the flow patterns is characterized by high turbulence intensity and complex velocity fields frequently imposed by geometry of the intakes, diversion channels or tunnels and of the energy dissipaters.

Recent advances in computational and hardware technology allow employing numerical solution for the analysis of flow through hydraulic structures. For this reason, the dam engineering community started recently to make use of CFD models, namely in spillways and water intakes ([1]; [2]; [3]; [4]; [5]).

However, although physical models can translate the complexity of flows in hydraulic structures, they are more expensive and time-consuming than numerical models. Furthermore, 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 through an integrated approach may mitigate shortcomings of both.

The primary objective of this paper is to present a combined approach, including CFD numerical and scale physical modelling, to address the behavior of the spillway of a large dam, to be constructed in the Mondego river, as a part of the Portuguese National Program for Dams with High Hydroelectric Potential (PNBEPH).

Eptisa and JESÚS GRANELL Ing. Consultores developed the construction project and conducted the CFD spillway modelling for the company Endesa Generación. The National Laboratory of Civil Engineering (LNEC) performed the study in the physical scale model.

The general characteristics of the dam are presented in section 2 and the physical model is described in section 3. The characteristics of the CFD numerical model and the main results are presented in sections 4 and 5. Finally, conclusions are drawn in section 6.

2.-GENERAL CHARACTERISTICS OF THE PROJECT

The spillway designed passes through the right abutment of the arch dam of 106 m high. The magnitude of the flow rates, their position crossing the abutment and their orientation determine the project solution described hereafter. It is a chute type spillway consisting of an entrance channel, a control structure, a discharge channel and a terminal structure, bucket type (Figure 1).

The spillway for the standard project flood was designed for a return period of 10,000 years assuming that the duration of rainfall is the same of the time of concentration of the basin, generating a design flow of the spillway of 1,648.5 m^3/s (design flow). For a return period of 10,000 years with a rainfall duration twice the time of concentration of the basin the design flood of the spillway would be of 1,885.4 m^3/s (extreme flood). Both of them generate unitary elevated flow rates in the control section of the spillway: 57.84 $m^3/s/m$ and of 66.15 $m^3/s/m$.

The abutment holds the strengths of the arch and transmits them to the foundation, with allowable stresses. Furthermore, the orientation of the abutment, tangent to the guideline of the vault at the start of the arches, predetermines the direction of the discharge channel (perpendicular to the abutment) making the water feed from the reservoir more difficult.

Since a spillway controlled by gates is necessary, the designed solution is a spillway with frontal feeding, although the water section over the abutment rotates (using a curve) with respect to the discharge channel in order to be oriented to the reservoir.

The weir comprises 3 separate spans of 9.5 m width where there are 3 Taintor gates of 9.5 x 6.8 m. The concrete lip of the spillway is placed at 294.00 m and the gates have their crowns at 300.00 m.

Immediately after the gates there is a transition curve which guides the discharge to bring it into the abutment perpendicular, there is also a narrowing of the spans, from 9.5 m, in the mouth, to 6.5 m, along the abutment.

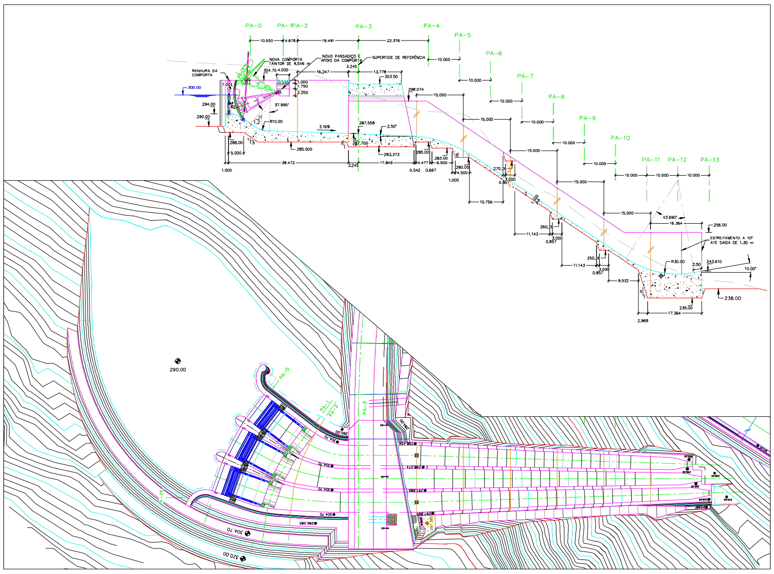


Figure 1. Spillway designed.

The three spans cross the abutment by means of the same number of recesses. These recesses have trunk shape: rectangular at the bottom with 6.5 m width and domed in their upper side (half ellipse).

The three spans that form the spillway remain independent from each other and are separated by two continuous piles of 1 m width. For this reason their slopes are different as they have to be adapted to the hillside, helping to reduce excavations and concrete fillings.

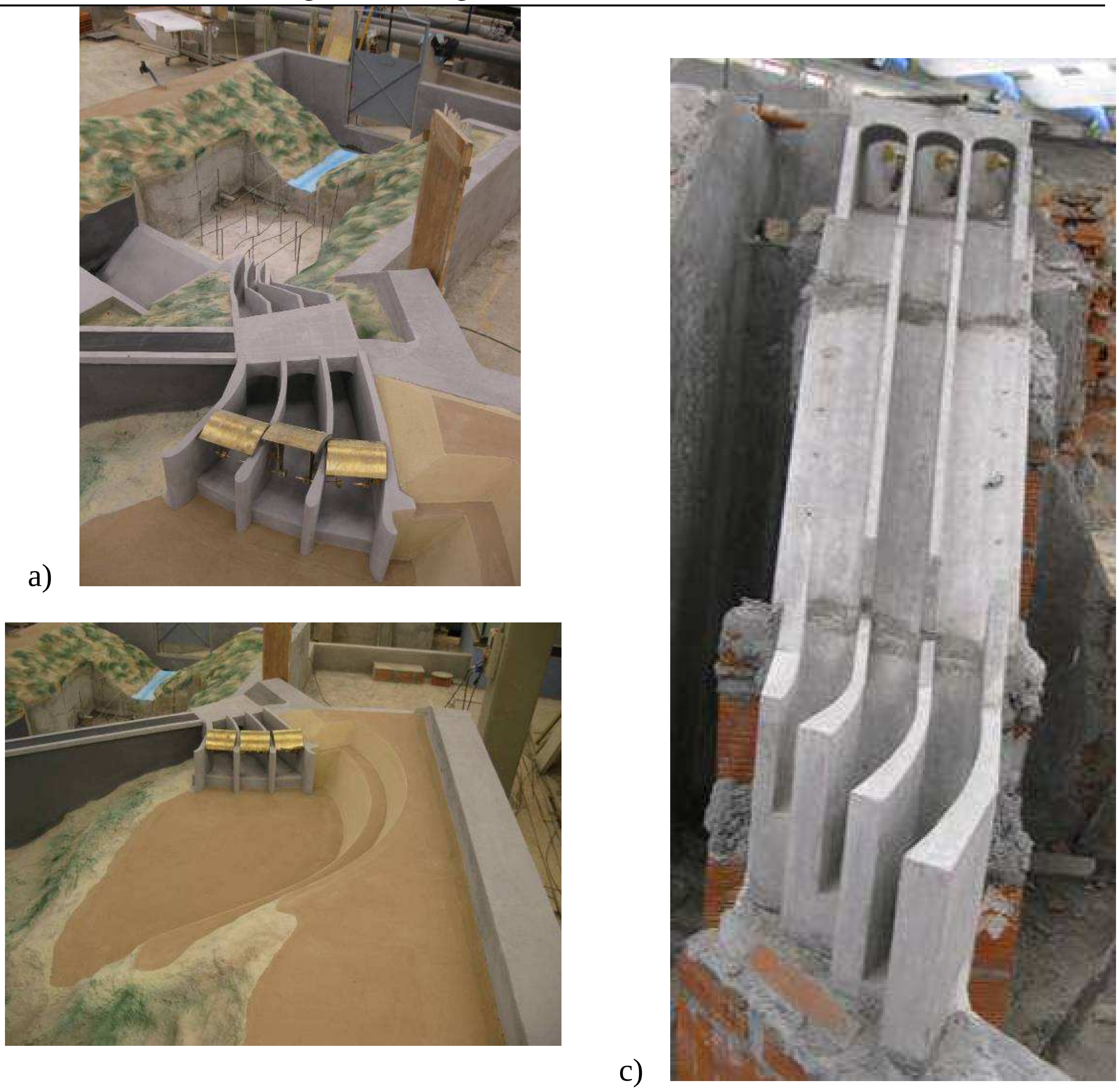
In elevation, the slopes start in escarpments, defined by Bradley profiles (parabola grade 1.85) for design nappe of 8 m, to which almost immediately follows a hollow in the shape of a circle in each span, up to the flat slope with which the channels cross the abutment, with an approximate slope of 4%. Downstream the flat slope described, there is a crest shaped as a second degree parabola till the next straight alignment, with an approximate slope of 66%.

In plan, downstream of the abutment, the buckets of the sides are convergent, for this reason, the overall channel width in the beginning of this section is 23.50 m (three spans of 6.50 m, plus 4 piers of 1 m each one); near the bucket, the total width is 15.69 m. The three vans distribute their width equitably all the way.

Regarding the bucket, there have been different configurations that allow to compute the best location of the flow impact in the riverbed and in the best trajectory of the jet. At present, the last configuration is being tested in the physical model to be evaluated: it is the "horsetail throwing" which generates a vertical expansion of the jet, an increase of air emulsion and a reduction of the impact pressures.

3 PHYSICAL MODEL

The study was performed based in a 1:60 scaled physical model. The model represents the right half of the dam and the chute spillway in its right abutment (Figure 2). The model includes 200 m of the reservoir, upstream of the crest dam, and a 500 m long river reach downstream ([6]; [7]).



a) b) c)

Figure 2: Physical model. a) general downstream view; b) view of the entrance channel; c) view of the chute and terminal structure

Flow, pressures and water levels are measured in the scaled physical model with the following equipments:

- gauge limnimeters, which errors are considered to be inferior to ±0.2 mm, to measure the reservoir water surface elevations and tailwater elevations;
- electromagnetic flowmeters, which measuring uncertainty is less than 1%, to measure discharges;
- piezometers to measure average pressures.

4 NUMERICAL MODEL

FLOW-3D® is CFD (Computational Fluid Dynamics) software was originally developed by the Los Alamos National Laboratory in New Mexico and further developed, for over 30 years, by FlowScience Inc.

To solve the Navier - Stokes equations, the model uses the Finite Volume Method (FVM). This software is especially useful in complex geometries and water free surface calculations since, besides the FVM, it has two algorithms [FAVOR ® and TruVOF®] that represent the elements and the free surface, respectively.

By integrating geometry into conservation equations, the FAVOR® algorithm (Fractional Areas/Volumes Obstacle Representation) models the solids inside each control volume. Through the design of a structured mesh, the accurate geometry of the solids is obtained, without having to localize the mesh nodes on their surface.

The TruVOF® (Volume of Fluid) algorithm is used to represent the interface between two fluids, when this interface is present in the calculations. Thanks to this algorithm, the software is able to correctly localize and indicate the direction of the free surface and follow its movement precisely. Besides, what is more important, the algorithm allows to impose the boundary conditions on the free surface, avoiding both calculations in the air phase and the diffusion of the interface.

In the particular case of the spillway studied, the complete spillway has been modelled as also part of the reservoir and of the river downstream of the dam (Figure 3). The general cube-shaped net used in the spillway area was 0.5 m and was increasing its size towards the edges until 2 m. Simulations have been carried out to study certain areas as the curve in the beginning of the rapid, starting by nearly steady re-netting till 0.125 m to restart. This action provided models of between 2 and 21 million of active cells.

Maintaining the level in the reservoir which matches with the flow rate to be tested and the river level for that flow rate were the boundary conditions defined for the outline.

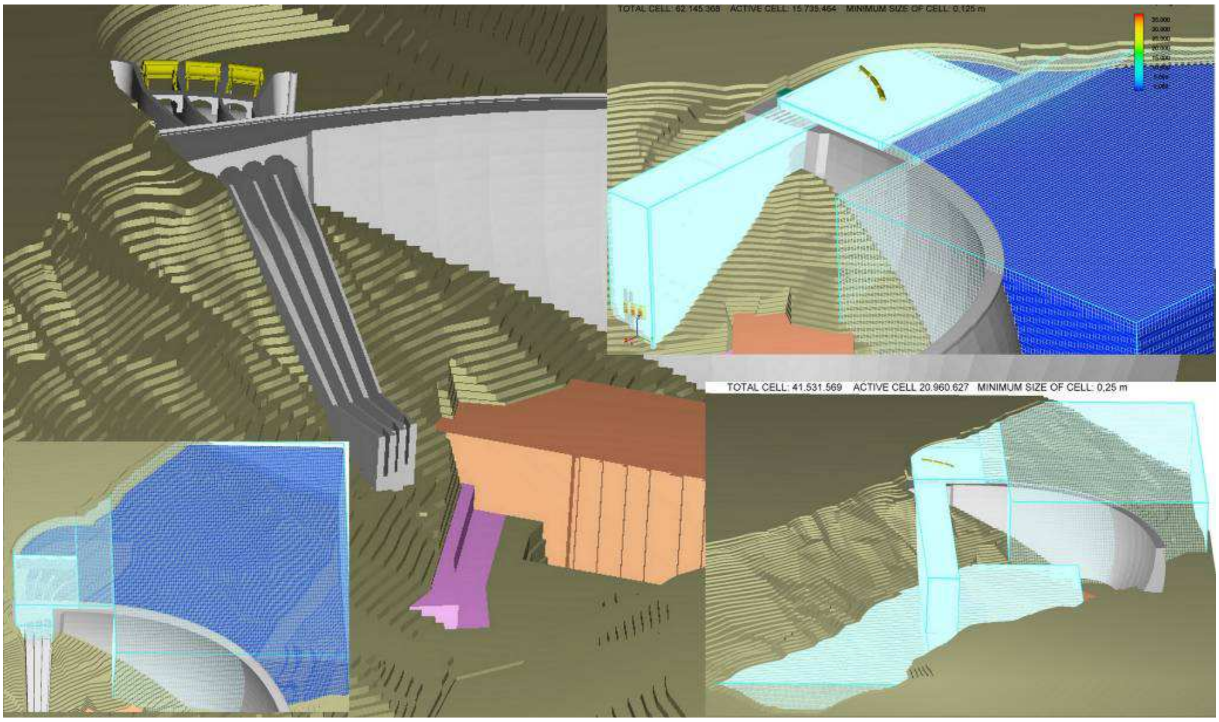


Figure 3. Spillway CFD model

5 COMPARISON OF FLOW IN NUMERICAL AND PHYSICAL MODELS

5.1 Entrance

Solution 1 of the entrance

In the first solution studied there was special concern in avoiding high speed in the flow on the curve in order to avoid superelevations. This fact, as evidenced in the numerical and physical models, led to a misdevelopment of the rapidness in the initial part of the rapid and therefore the control section has not been adequately fixed in the spillway crest. The flows obtained were below expectations.

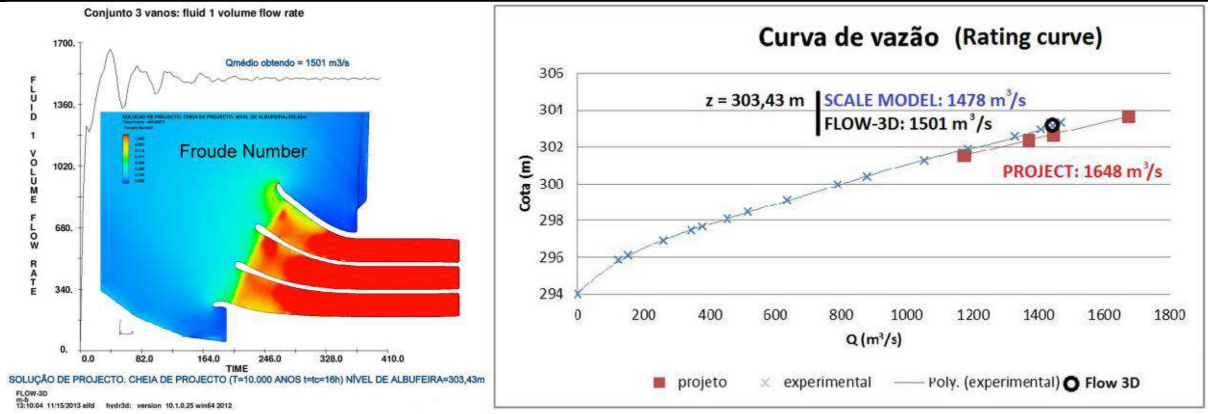


Figure 4. Solution number one of the entrance. Hydraulic capacity.

In the left side of Figure 4, the computed Froude numbers in the entrance of the spillway are presented (in red Froude ≥1, in blue Froude <1) from the numerical model. It can be seen that in the beginning of the rapid and almost up to the curve the regimen is slow. In the right side of Figure 4 the experimental and numerical rating curves are presented; it can be seen that the spillway discharges (computed and tested) are lower than expected.

Solution 2 of the entrance

In order to increase the hydraulic capacity up to the necessary values, the initial chute slope was modified as follows (Figure 5):

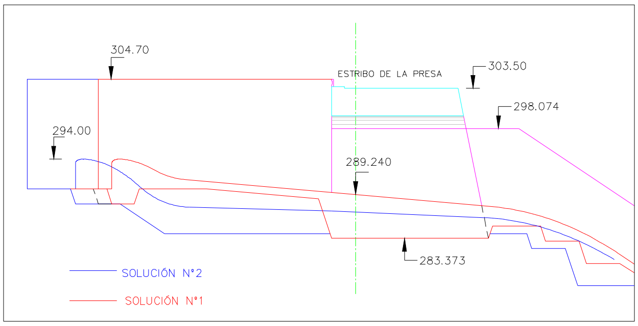


Figure 5. Solution number 1 and number 2.

The result was the expected one, an increase of the spillway hydraulic capacity, reflected in the results of both the numerical and the physical model. However, the increase of the speeds in the curve caused various undesirable effects for the extreme flows like the flow super-elevation in the abutment area (commented hereinafter). Therefore, the axis of the gate of the exterior span got wet, occasionally for the design flood, and, almost permanently, for the extreme flood (Figure 6).

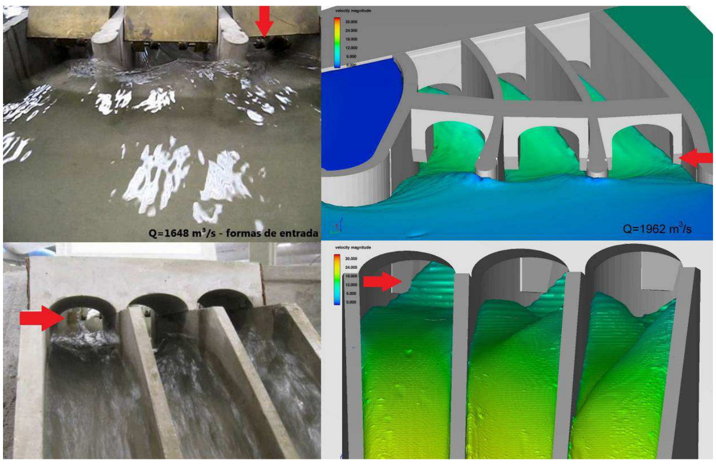


Figure 6. Solution number 2 of the entrance

Solution 3 of the entrance

The new solution, is still pending for evaluation of the physical model, and differs from the one above as following (Figure 7):

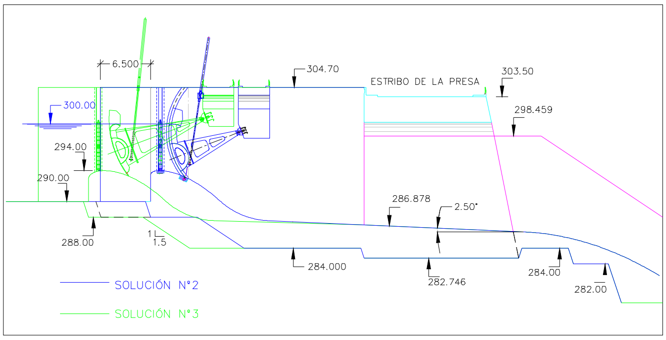


Figure 7. Solution number 2 and number 3.

As shown in Figure 7, the threshold position and the gate are advanced, the radius of the gate is increased, the position of the axis moves upward up to 1.7 m and the supporting structure of the axis stays out of the curve.

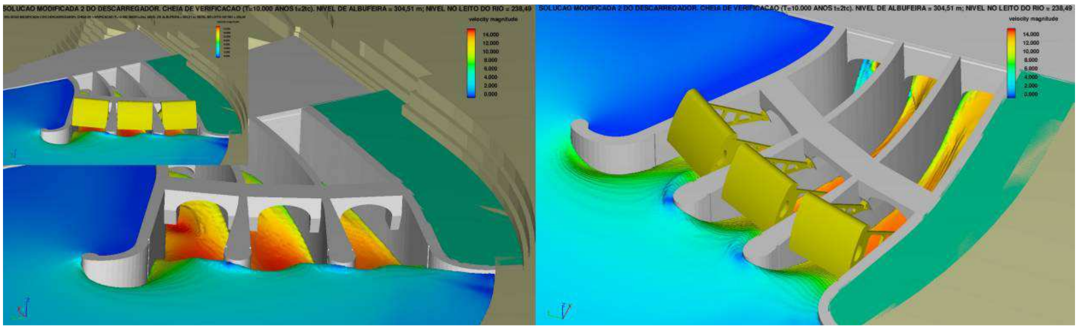


Figure 8. Solution number 3. Entrance

According to the numerical model results, the gate axis remains out of reach of water, even for the extreme scenario. The effects of the flow passing through the curve are analyzed in the next section.

5.2 Abutment dam passage

Solution 1 passing through the abutment

Figure 9 presents the flow aspect both in the numerical and physical models.

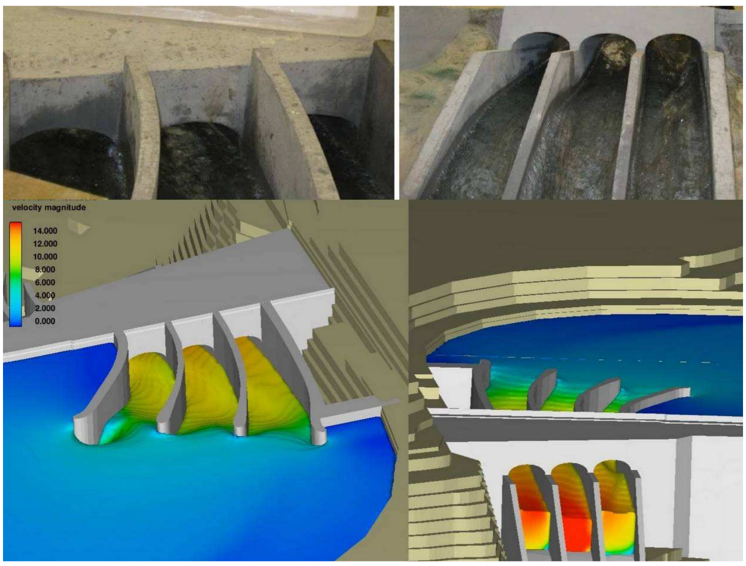


Figure 9. Solution number 1. Pass through the abutment. Super-elevations of the nappe

The behaviour is reasonably adequate. Flow super-elevations due to the curve are not relevant due to the moderate velocity (order of 10 m/s). However this solution was invalidated since it did not provide the necessary hydraulic capacity.

Solution 2 passing through the abutment

Velocities in the curve passage increase substantively with respect to solution 1. In this case, the velocities are of around 13 m/s, leading to an increase of flow super-elevation in the curve passage and to a rebound effect in the next section which is transmitted to part of the chute discharge (Figure 10). The interior passage of the abutment suffer the superelevation caused by the curve passage and the first rebound. In both cases, for the extreme flood, there is a risk of turning over the nappe inside of the abutment.

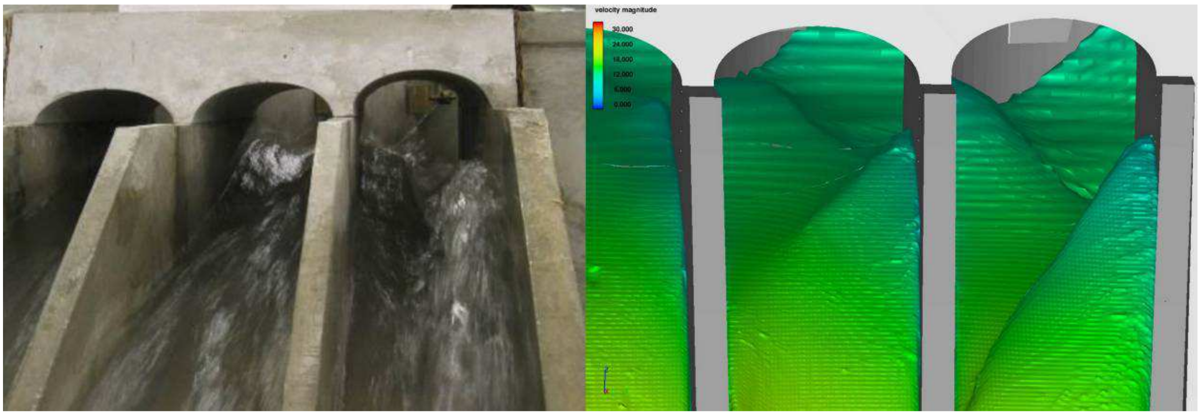


Figure 10. Solution number 2. Pass through the abutment. Super-elevations and rebound effects of the nappe

Solution 3 passing through the abutment

This solution reduces substantially the velocity with respect to solution 2. In this case, flow velocities are of the order of 11 m/s, reducing the super-elevation in the curve passage and the other problems associated to the discharge (Figure 11).

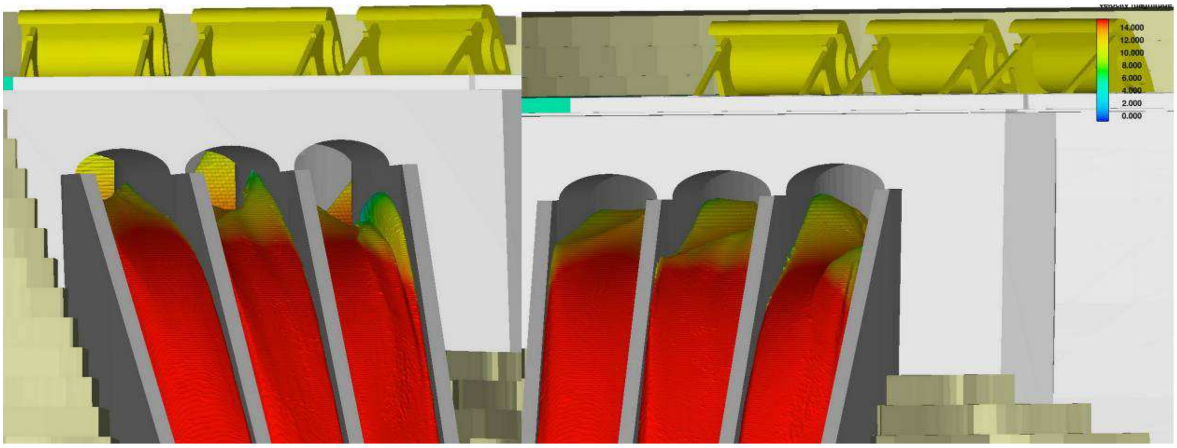


Figure 11. Solution number 3. Pass through the abutment. Super-elevations and rebound effects of the nappe

5.3 Chute

Solution 1 of the chute

The behaviour is adequate given that the hydraulic problems associated to the curve have been reduced. The effects of the convergence are not substantial and do not produce crossed waves when arriving the bucket (Figure 12).

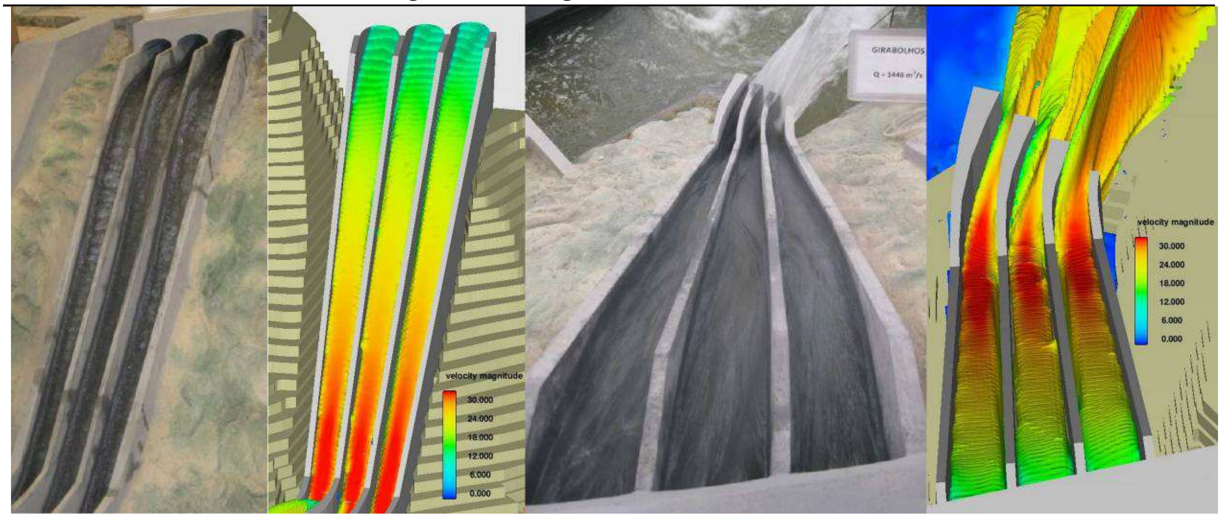


Figure 12. Chute. Solution number 1

Solution 2 of the chute

The rebound effect of the super-elevation in the curve passage spreads to the rapid discharge area, downstream curves zone. Evidence shows, in the numeric model as well as in the physical tests, that the channels wall heights are properly designed (Figure 13). In this respect, note that in the extreme scenario (1,885.4 m^3/s) the channel walls are, in some spots, sometimes overtopped.

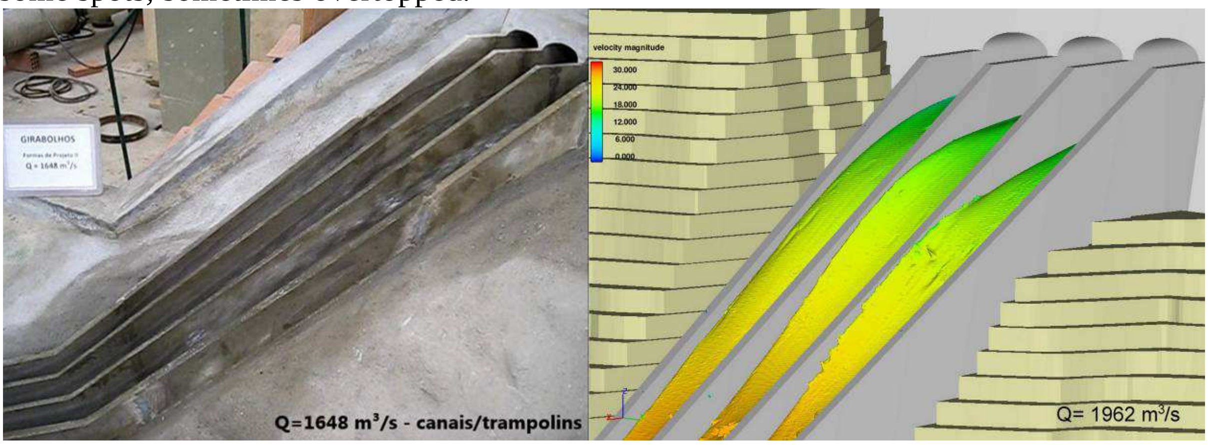


Figure 13. Chute. Solution number 2. Rebound effects extended.

Solution 3 of the chute

The effects mentioned in the previous solution are mitigated in this new configuration as it can be seen in the numeric model results.

Considering the entrance and the discharge capacity, and in the absence of the physical model tests, the current solution is reasonable because it provides the necessary hydraulic capacity according to the regulations. Furthermore, the effects of the curve at the initial part of chute are also reduced. This solution also does not affect the gate axis, avoiding the risk of turning over the nappe in the abutment (cut and cover tunnel) and limiting the propagation of the effects of flow super-elevation to the chute (Figure 14).

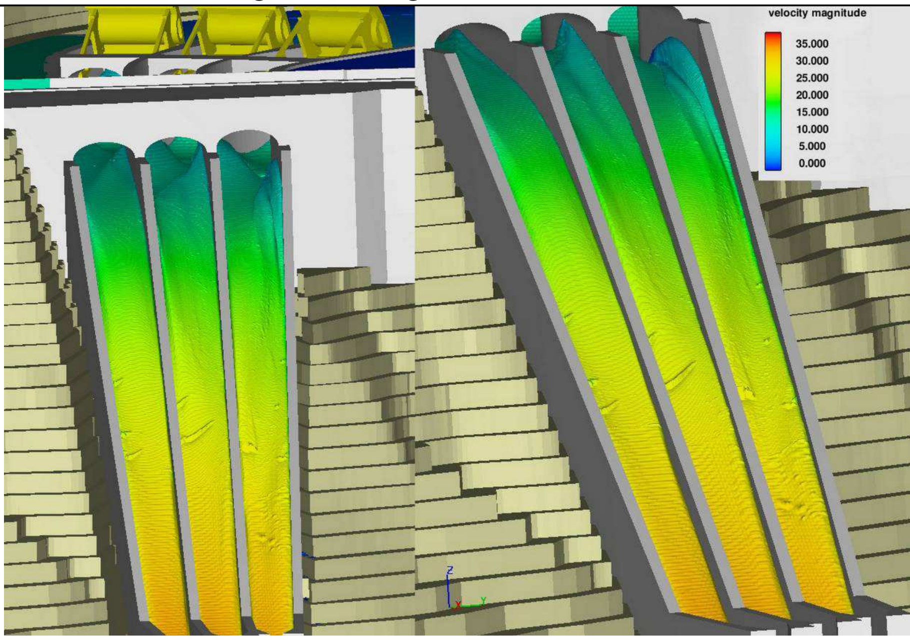


Figure 14. Chute. Solution number 3. Hydraulic behaviour improved respect solution 2.

5.4 Bucket

Solution 1 of the bucket

The first solution was a bucket, which as well as throwing, turns over the nappe in order to line up the flow with the riverbed. The numeric and the physical models one demonstrated that this solution was not appropriate, because the jet impacts too close to the hillside (Figure 15).

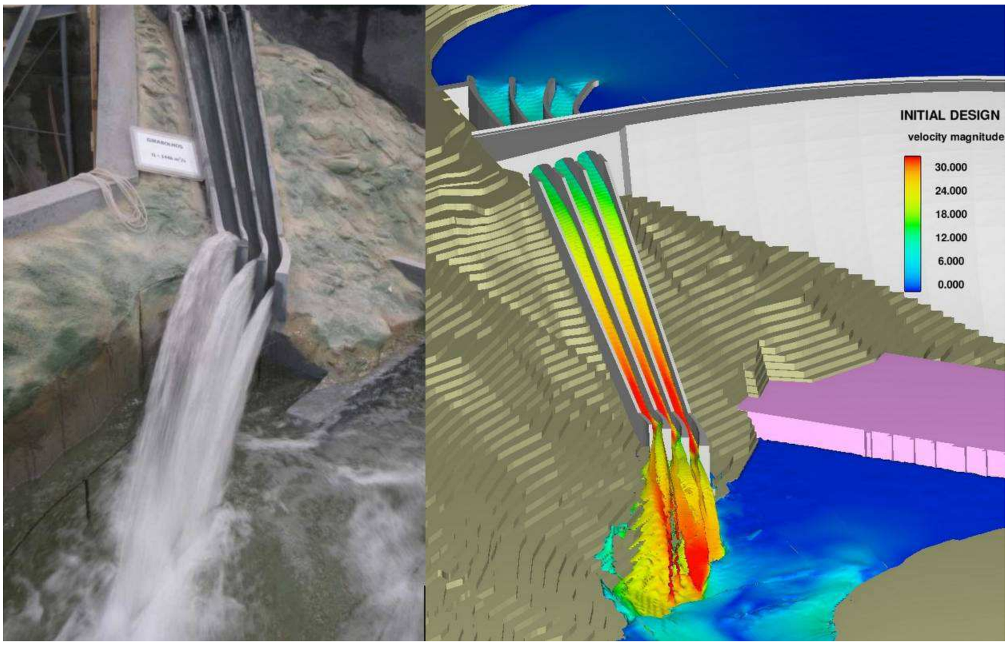


Figure 15. Solution 1 of the bucket

Solution 2 of the bucket

As a second solution, a cylinder bucket was studied, with angles of 10º (in the left and middle chute) and 12.5º (in the right chute).

Figure 16 presents the jet both in the numerical simulation as well as in the physical test.

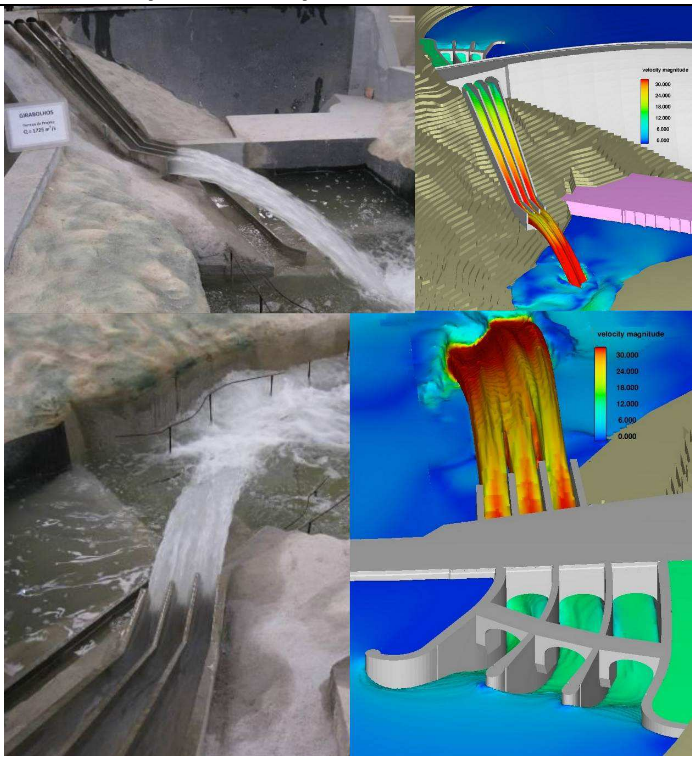


Figure 16. Solution 2 of the bucket

As seen in the pictures, the jet trajectory is adequate, taking into account that the impact occurs in the middle of the riverbed. However, the jet is compact so it does not help to air emulsion and, subsequently, this does not help to reduce flow velocity. These facts can be translated in a higher pressure impact. For this reason a 3rd solution was studied.

Solution 3 of the bucket ("horsetail").

The 3rd solution, which does not have physical model tests yet, consists in a abrupt narrowing in each span, in order to promote a vertical expansion of the nappe (in horsetail) and therefore a bigger air emulsion.

Figure 17 presents aspects of the numerical model results.

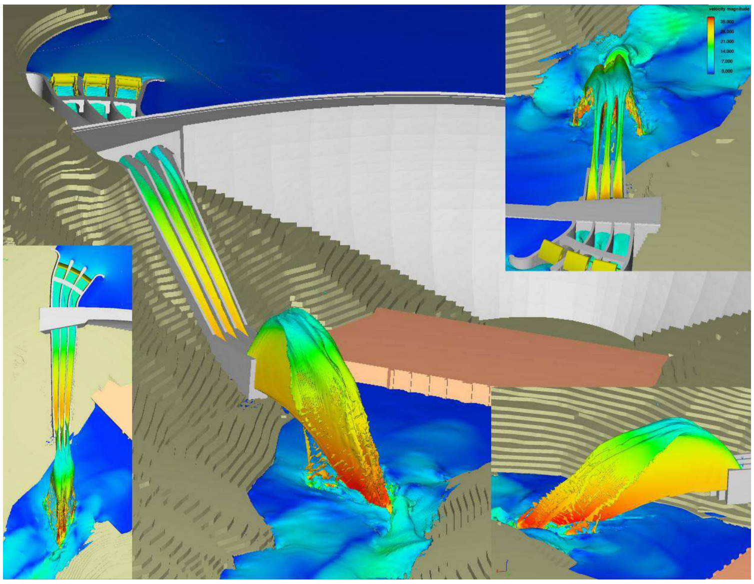


Figure 17. Solution 3 of the bucket

6 CONCLUSIONS

The studied spillway is characterized by the occurrence of elevated unitary flow rates, presenting various challenges to its design. By using Computational Fluid Dynamics (CFD) and physical models, it has been possible to adjust some project solutions in different issues, namely:

- the flow conditions in the entrance zone, particularly in the curved zone of the channels after the weir crests;
- the free surface development along the side walls of the chutes;
- the jets from the buckets.

The different solutions have been improving successively, both at numerical and physical models, the hydraulic behaviour of the spillway. The physical model of the last solution is being studied at the moment, however the numeric model shows the following conclusions:

- The hydraulic capacity of the spillway is adequate.
- The layout of the gates is proper.
- The transition curve does not generate unacceptable superelevations, even for the extreme flood.
- The convergence of the buckets in the chute does not produce adverse effects.
- A centered and "horsetail" bucket produces a significant air emulsion.

In conclusion, the use of CFD models was attractive, since, comparing to physical models, it allowed 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, the CFD models was validated for each spillway design solution. In this regard, the proximity between experimental and numerical data for the dam spillway chute contributed to the validation of the numerical model.

7 REFERENCES

[1] Higgs, J., and Frizell, K. W. ‘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).

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

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

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

[5] Meireles, I., Silva, S., Viseu, T., Sousa, V. and Dias da Silva, J.“Experimental and numerical study of water intakes: case study of the Foz Tua hydropower plant”. 3rd IARH Europe Congress with the theme “Water – Engineering and Research”. Porto, April, (2014).

[6] Viseu, T. “Barragem de Girabolhos. Estudo hidráulico em modelo reduzido do descarregador de cheias. Ensaios das formas de projeto” (in portuguese). Technical Note DHA/NRE 08/2013, October, (2013).

[7] Mendes, L. and Viseu, T. “Barragem de Girabolhos. Estudo hidráulico em modelo reduzido do descarregador de cheias. Ensaios das formas de projecto II” (in portuguese). Technical Note DHA/NRE 21/2015, January, (2015).