IAHR AIIH
INTERNATIONAL SYMPOSIUM ON HYDRAULIC STRUCTURES
CIUDAD GUAYANA, VENEZUELA, OCTOBER 2006

INTERACTIONS BETWEEN NUMERICAL AND PHYSICAL MODELLING FOR THE DESIGN AND OPTIMIZATION OF HYDRAULIC STRUCTURES EXAMPLE OF A LARGE HYDROELECTRIC COMPLEX

ERPICUM[1] S., ARCHAMBEAU[1] P., DEWALS[1,2] B.J., DETREMBLEUR[1] S., LEJEUNE[1] A. & PIROTTON[1] M.
[1] Laboratories of Fluid Mechanics, Applied Hydrodynamics and Hydraulic Constructions, Institute of Civil Engineering, University of Liège, Belgium, s.erpicum@ulg.ac.be, agh.lejeune@ulg.ac.be, michel.pirotton@ulg.ac.be
[2] FNRS Research Fellow

ABSTRACT

This paper describes the successful combination of both physical and numerical modelling techniques applied to the scale model study of a large hydroelectric complex in India. Thanks to accurate numerical simulations, the physical model layout and the time to studies have been drastically reduced.

First, prior to the design of the scale model, a global numerical simulation of the dam reservoir has been carried out, on the basis of the entire set of available topographic data. The simulation results supply the flow conditions in the upstream reservoir, particularly in the vicinity of the dam and the spillway crest.

Subsequently, a second numerical computation has been performed to simulate the flow in a scale model representing only a small part of the reservoir upstream of the dam. The suitable correspondence between flow conditions in both cases has been demonstrated.

This two-steps approach has enabled a significant reduction in the size of the scale model, and hence, a lower cost as well as a shorter delay for building the model. Simultaneously, it has been possible to decrease the scale factor at constant cost.

In conclusion, the present paper provides convincing evidence that the increase in computers performance and the efficiency of contemporary free-surface flow solvers lead to useful numerical information for the optimal design and exploitation of scale models.

KEY WORDS: numerical-physical interaction; scale model; finite volume

INTRODUCTION

In the field of hydraulic engineering, numerical models of great reliability and accuracy are today available for 1D, 2D as well as, to some extent, 3D simulations. WOLF software includes a series of such numerical tools for simulating a wide range of free surface flows and transport phenomena. It has been entirely developed at the University of Liège for almost ten years and is still continuously enhanced.

On the other hand, scale models have been recognized for a long time as powerful tools for the design of hydraulic structures. Provided that right similarity laws are used, they give reliable and uncontested results. Such experimental tests on scale models remain attractive for complex situations but they suffer intrinsic weakness such as significant cost and delay for building, as well as low geometrical flexibility once the model has been constructed.

In order to circumvent these disadvantages, numerical simulations more and more advantageously complement physical experimentations. Global numerical models are indeed useful to circumscribe the hydraulic problems to be solved with detailed physical models at a greater scale. This philosophy has been adopted for years at the University of Liège, where the quasi-three-dimensional flow solver WOLF2D is used in the Laboratory of Hydraulic Constructions in interaction with physical modelling (Pirotton *& al.*, 2003).

The present paper depicts an example of this successful combination of both physical and numerical modelling through the description of a large dam in India. Thanks to accurate numerical simulations, the physical model layout and the time for the studies have been drastically reduced.

After a brief presentation of the main features of WOLF software package, with a focus on WOLF2D, the characteristics of the hydroelectric complex to be studied are depicted. Available data are described and the boundary conditions to be imposed for numerical and physical modeling are explained. Then, the results of the flow computation in the whole reservoir are exposed and identification of a proper scale model geometry is deduced and justified. Finally, the validity of the selected scale model extent and geometry is demonstrated, prior to a short discussion of the results of the subsequent experimental tests.

WOLF SOFTWARE PACKAGE

The free surface flow computation package "WOLF" provides, in the same environment, the resolution of the 1D Saint-Venant equations, the 2D shallow-water equations as well as a process-oriented hydrological model and powerful optimization capabilities based on Genetic Algorithms (Fig. 1). Hence, the components of WOLF allow the package to deal with all free surface hydraulic phenomena, from hydrological runoff and river propagation (Archambeau *& al.*, 2001) to extreme erosive flows on realistic mobile topography, such as gradual dam breaching processes (Dewals *& al.*, 2002).

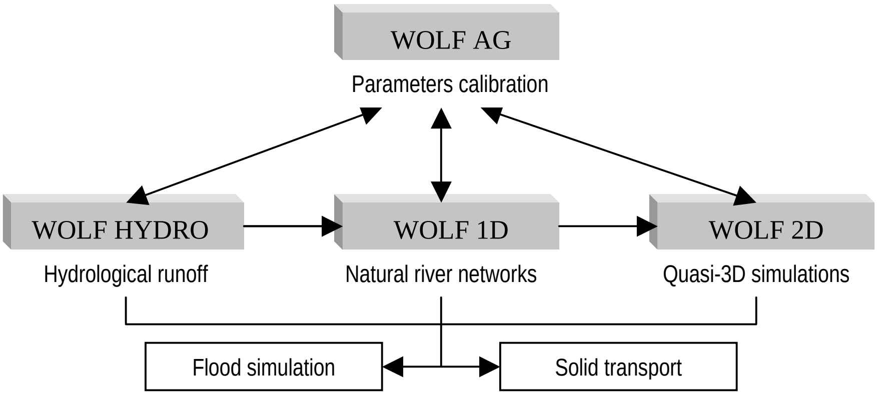

Figure 1.- General organization of WOLF computation units.

Each code handles general multiblocks meshes, dealing with natural topography and mobile bed simultaneously, for any unsteady situation with mixed regimes and moving hydraulic jumps.

The validation of WOLF has been performed continuously by comparisons with analytical solutions as well as field and laboratory measurements.

The interactive and unique user-interface, with high performance pre- and post-processing, allows monitoring 3-D large-scale runs graphically while they proceed, as well as powerful visualization features (such as generating 3D videos).

WOLF2D

The two-dimensional multiblock flow solver WOLF2D, used in the present study, solves the conservative form of the so-called shallow water equations (SWE):

$$\frac{\partial h}{\partial t}+\frac{\partial uh}{\partial x}+\frac{\partial vh}{\partial y}=0 \qquad [1]$$

$$\frac{\partial}{\partial t}(hu)+\frac{\partial}{\partial x}\left(hu^2+\frac{gh^2\sin\theta_z}{2}\right)+\frac{\partial}{\partial y}(huv)=-gh\sin\theta_z\frac{\partial z_b}{\partial x}+ghJ_x\cos\theta+gh\sin\theta_x \qquad [2]$$

$$\frac{\partial}{\partial t}(hv)+\frac{\partial}{\partial y}\left(hv^2+\frac{gh^2\sin\theta_z}{2}\right)+\frac{\partial}{\partial x}(huv)=-gh\sin\theta_z\frac{\partial z_b}{\partial y}+ghJ_y\cos\theta+gh\sin\theta_y \qquad [3]$$

where h is the water height, u and v the velocity components, z_b the bottom elevation and J_x and J_y the components along axis x and y of the friction slope. The angles θ_x, θ_y and θ_z quantify the inclination between the vertical direction and, respectively, each axis of reference x, y and z.

The main assumption behind this set of equations is that the velocity component w normal to the mean flow plane remains weak compared to the velocities u and v in the main flow plane, so that the square of their ratio may be neglected. On the contrary, no assumption on the bottom inclination is needed to set up the SWE. So, steep slopes are easily modeled by the angles θ_x, θ_y and θ_z enabling a proper definition of the x- and y-axis of reference following locally the mean bottom slope (Pirotton, 1994).

Extra terms or extended sets of equations are implemented in WOLD2D to model additional phenomena, such as turbulence processes, fluid-structure interaction due to moving bodies in the fluid flow (Erpicum *& al.*, 2006), bottom curvature (Dewals *& al.*, 2006) as well as air or sediments transport (Dewals *& al.*, 2004).

The space discretization of the equations is performed by a widely used finite volume method. This ensures the mass and momentum properties to be conserved, even across discontinuities such as hydraulic jumps. Flux treatment is based on an original flux-vector splitting technique developed for WOLF. Fluxes are split according to the sign of the flow path, requiring a suitable downstream or upstream reconstruction for both parts of the convective term according to a stability analysis. Efficiency, simplicity and low computational cost are the main advantages of this scheme. Variable reconstruction can be selected to gain first or second order accuracy on regular grids. Besides, an explicit Runge-Kutta scheme or an implicit algorithm is applied to solve the ordinary differential equation operator.

WOLF2D includes an efficient mesh generator and deals with multiblocks structured grids. These features increase the size of potential problems to be solved and allow mesh refinement close to interesting areas without leading to prohibitive CPU times (Erpicum *& al.*, 2004a).

The algorithm is designed to deal automatically with any moving boundary. It incorporates an original method to handle covered and uncovered (wet and dry) cells while strictly guaranteeing volume conservation (Erpicum *& al.*, 2004a). In addition, an adaptable grid extension technique achieves a drastic reduction in computation time, by restricting the simulation domain to the wet cells (and a surrounding narrow strip).

CASE STUDY: A MAJOR HYDROELECTRIC COMPLEX

The Laboratories have been entrusted by Electricité de France (EDF-CIH) to carry out studies on hydraulic scale model of a large 4 x 200 MW hydropower plant project in India. The project involves the construction of a large embankment dam (163 m high) with a 500 m long crest, located in a narrow valley of a highly erosive mountainous region. The spillway has to be designed for a 16,500 m³/s probable maximum flood (PMF) and the project design flood was 11,400 m³/s.

Five different scale models have been built to analyze the whole hydraulic behaviour of the plant. The first two models were focused on the sedimentation processes in the settling structure located upstream of the water intake (Erpicum *& al.*, 2004b). The third one represented the water intake and the four penstocks to the power plant, while the fourth one was related to the temporary diversion tunnels and the bottom outlet. Finally, the fifth model represented the whole complex, including the release structures (spillway) and part of the downstream valley. Among others, the flow conditions upstream and downstream of the dam had to be studied on this last model, and scour risks downstream had to be evaluated. The following sections will focus on the design of this fifth "global model".

Because of the meandering shape of the 40 km long valley upstream of the dam, it was difficult to assess *a priori* the flow conditions in the reservoir and near the spillway. Thus, the layout of the basin necessary to build a reliable scale model couldn't be easily determined.

Numerical simulations have thus been realized with WOLF2D, first, to determine the flow conditions in a large part of the upstream reservoir and, secondly, to find the most suitable layout and features of the scale model basin, in order to correctly represent flow patterns in the forebay and upstream of the spillway.

DATA AND BOUNDARY CONDITIONS

The Digital Elevation Model (DEM) of the upstream valley and the dam area has been supplied by EDF. This topography has been interpolated on a regular grid. Square meshes 3 meters in side have been used, achieving so a suitable compromise between the number of calculation cells and the proper representation of the structures details such as the piers of the spillway. In a second time, the dam, forebay, settling structure, water intake and spillway geometries have been added manually on the DEM, using the graphic interface of WOLF.

Figures 2 and 3 show three-dimensional views of the resulting DEM. A large meander of the river can be seen upstream of the embankment dam location. The spillway forebay is located on the left side of the river.

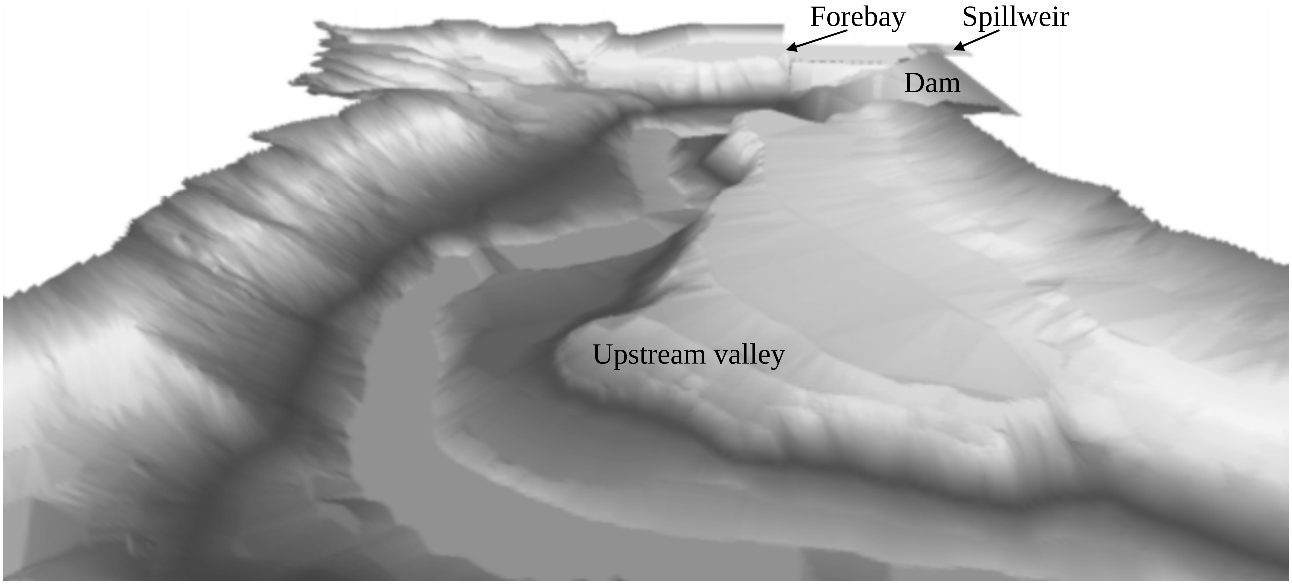

Figure 2.- Gobal DEM of the dam and upstream valley.

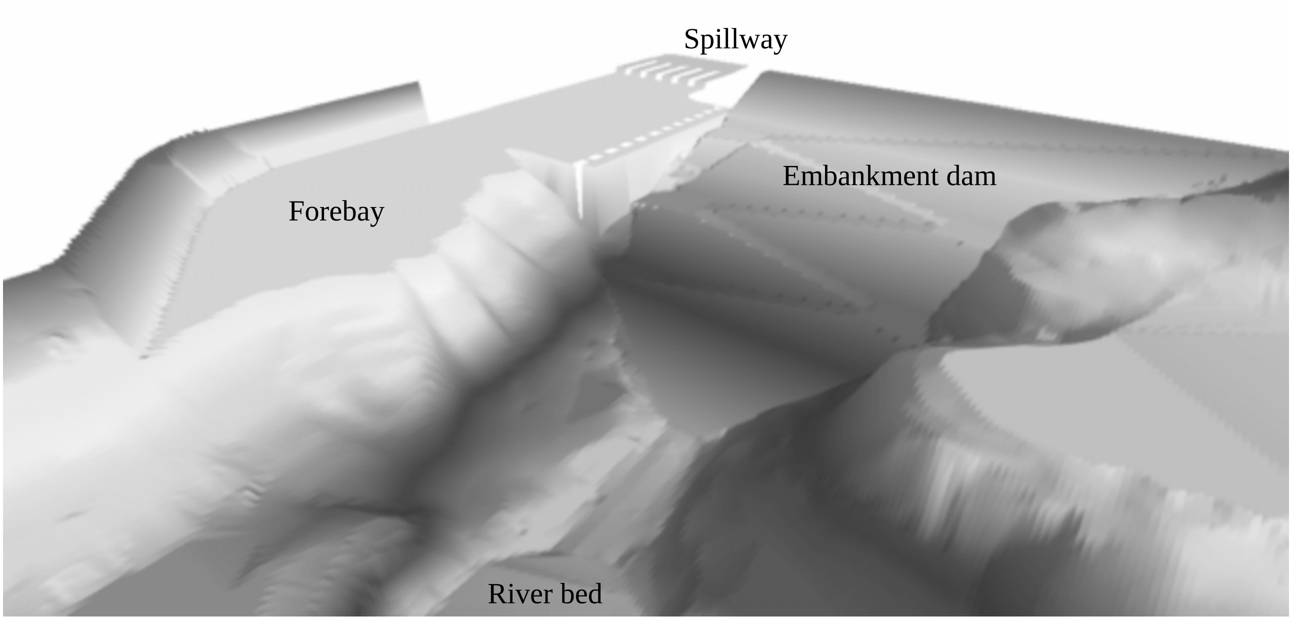

Figure 3.- Detail of the DEM near the dam.

The spillway topography has not been represented as a bottom topography but by means of the angles θ_x, θ_y and θ_z. In this case, the axis inclination has been made in a gradual manner on a length of 60 m downstream of the spillweir. A very steep slope (60 °) has been added to the end of the spillway to ensure a supercritical flow and thus to avoid the need for a downstream boundary condition.

All the simulations have been carried out with the PMF discharge of 16,500 m³/s. It has been injected at the upstream end of the simulation domain by using an infiltration technique to avoid biasing flow directions. Moreover, the discharge injection is adjusted at each time step as a function of the local water heights on each mesh to create a uniform flow velocity field.

As the transitions between emerged and submerged zones are automatically computed by the software WOLF2D, no boundary condition has to be imposed in the simulation.

FLOW CONDITIONS IN THE WHOLE RESERVOIR

In a first time, a global simulation of the reservoir, using the whole of the available topographic data, has been realized. It allowed determining reliable reference flow conditions in the upstream reservoir and near the release structures, before the simulation of smaller area representing the scale model upstream basin.

More than 178,000 finite volumes have been used for this first simulation to model a $1.6\ 10^6$ m² real area.

The results are illustrated on Figure 4 in terms of a distribution of specific discharge modulus (m²/s). The flow in the reservoir is divided into two main currents: one of them follows the river minor bed, where the friction resistance is smallest; while the other one cuts the bend of the river to directly feed the spillway by the shortest way. Between the two currents, a recirculation area can be observed. These specificities of the flow in the reservoir have to be taken into account when delimiting the scale model geometry to represent in a reliable manner the spillway alimentation conditions.

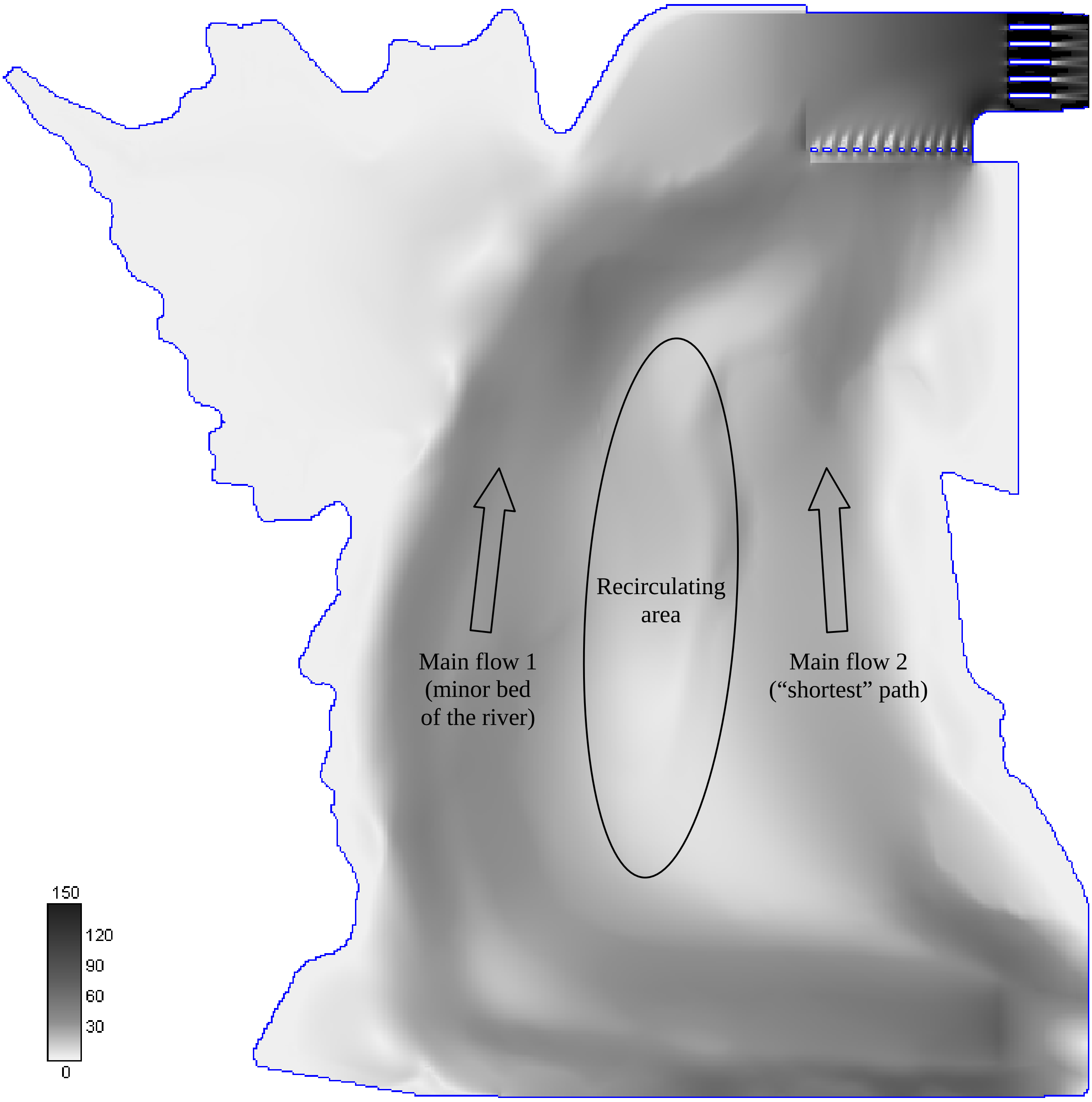

Figure 4.- Specific discharge modulus (m²/s) in the whole reservoir at PMF.

Figure 4 shows the spillway is preferentially fed by flows coming from the right bank. The discharge coming through the forebay is very small. These results suggest the forebay geometry could probably be reduced.

This is another advantage of the numerical simulations. They can suggest design modifications before the scale model construction.

SCALE MODEL UPSTREAM BASIN LAYOUT

The first geometry designed for the scale model upstream basin (black lines on Figure 5) included the end of the two main currents observed in the global simulation. To reliably feed the spillway on the scale model, these two currents had to be represented by the water feeding disposal, which was quite difficult to realize.

Finally, a smaller basin geometry, with the upstream wall located where the two currents join (gray lines on Figure 5), has been modeled. In this way, representing the real inflow conditions becomes simpler. The flow is almost uniform along the feeding wall and is mainly influenced by the topography, which can be easily reproduced on the scale model.

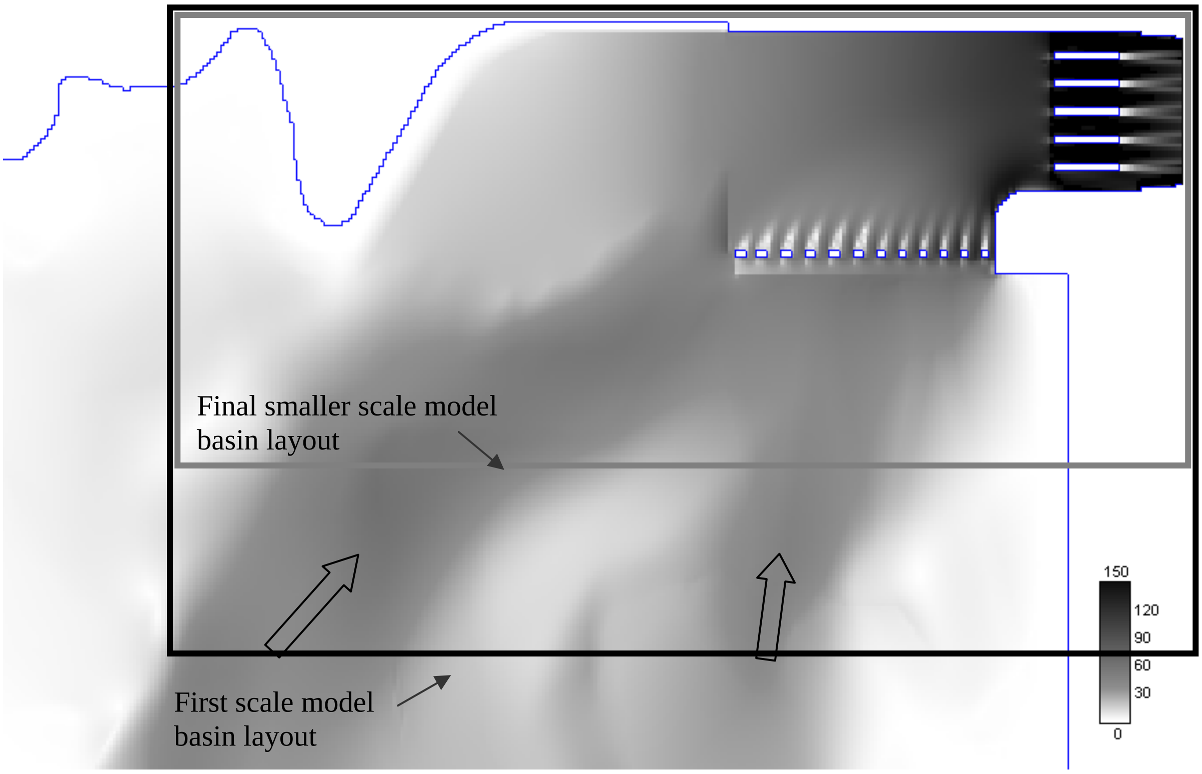

Figure 5.- Specific discharge modulus (m²/s) near the spillway at PMF – Layout of the scale model basins.

FLOW CONDITIONS IN THE SCALE MODEL

To validate the choice of the smallest basin layout, the flow pattern in the scale model must be proved to be satisfactorily similar to the flow pattern in the whole real reservoir. A simulation of the flow conditions in the scale model has thus been carried out with the same discharge as for the whole reservoir computation.

In the physical scale model, the alimentation area is delimited by a wall made of perforated bricks placed against the scaled topography. The water is injected behind the wall, and feeds the scale model by passing through the bricks, with a discharge proportional to the available water height. To represent this discharge repartition in the numerical model, the Figure 6 shows the used DEM, with a constant longitudinal topography behind the perforated wall, in the infiltration area.

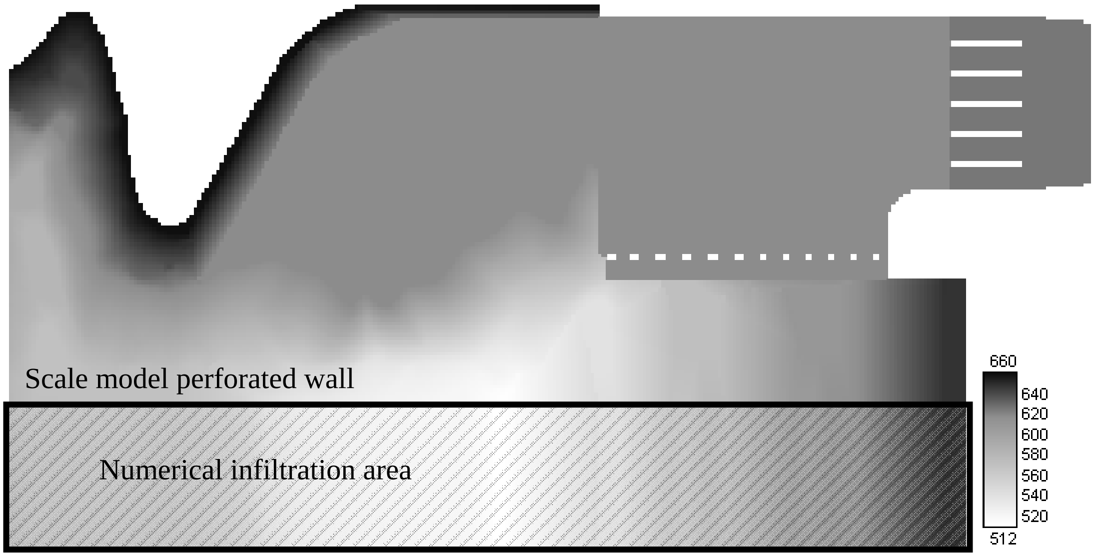

Figure 6.- Topography for the simulation of the reduced scale model basin

Figure 7 shows the results of the simulation in terms of specific discharge modulus distribution.

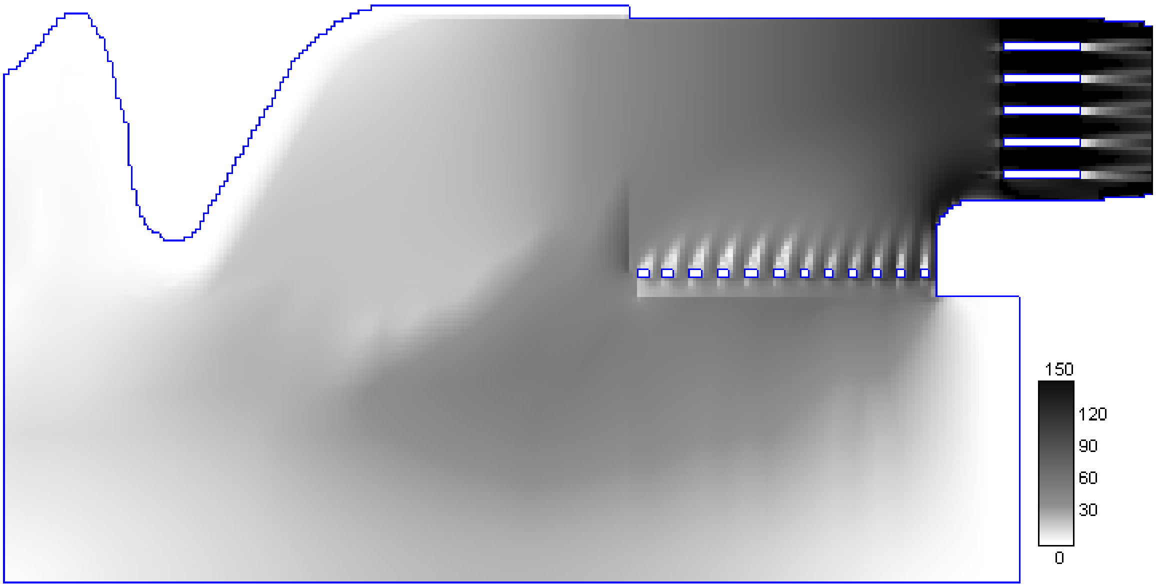

Figure 7.- Specific discharge modulus (m²/s) at PMF in the scale model upstream basin

RESULTS COMPARISON

The results of the two numerical simulations have been compared in four sections (Fig. 8) on the spillweir and near the forebay.

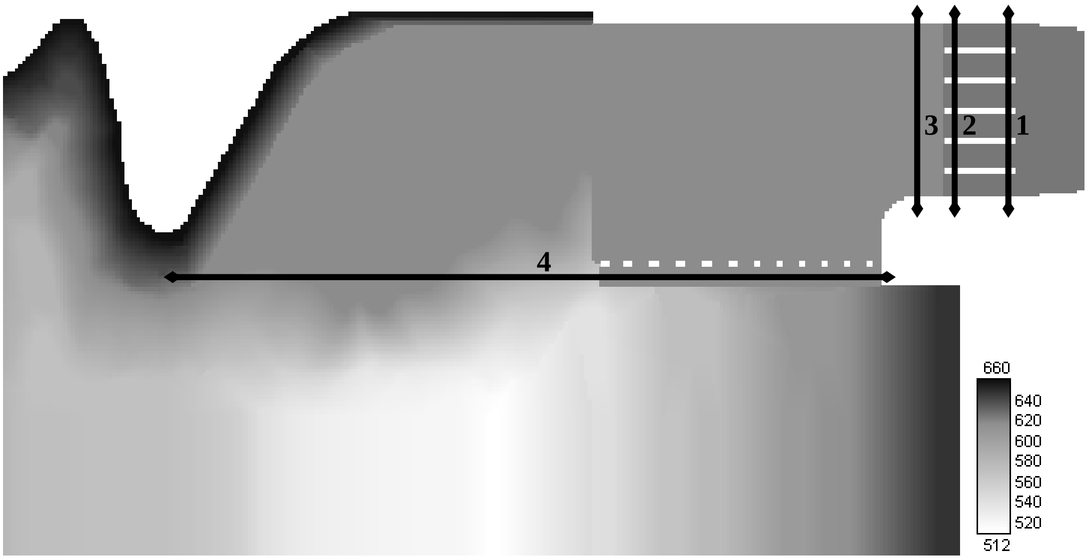

Figure 8.- Location of sections for results comparison

The concordance between the flow patterns in both cases is very good, for example regarding the discharge repartition in the approach area (Section 4 - Fig. 9). The differences between discharge modulus values remain very limited, and the main currents are located at the same place in both cases. The same may be concluded on the spillweir (Sections 1 to 3 - Fig. 10). At these last sections, the spillweir draught is so strong that no effect of the upstream feeding conditions can be seen anymore.

The good representativeness of the proposed scale model geometry in comparison with the global simulation can thus been concluded

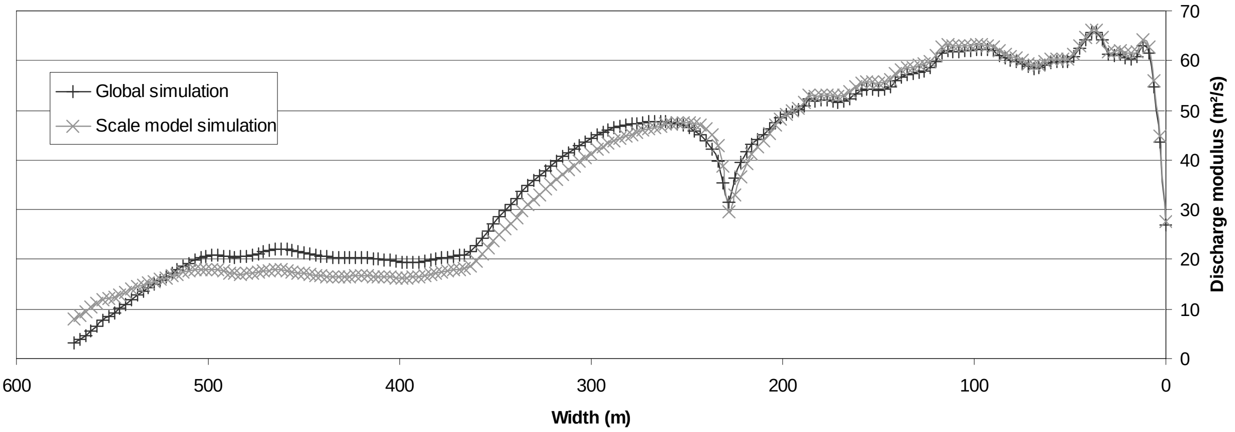

Figure 9.- Discharge modulus (m²/s) repartition on section 4

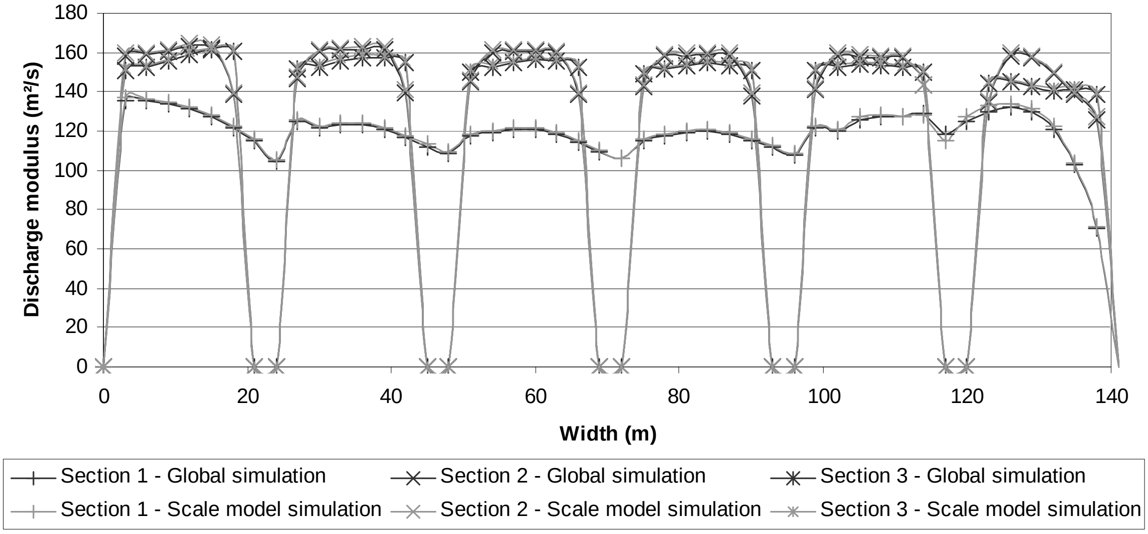

Figure 10.- Discharge modulus (m²/s) repartition on sections 1 to 3 on the spillweir

SCALE MODEL

The scale model has finally been built with a scale of 1:100 and with an upstream basin layout corresponding to those of the numerical simulations (Fig. 11). Flow pattern measurements during the model exploitation confirmed the numerical simulation results.

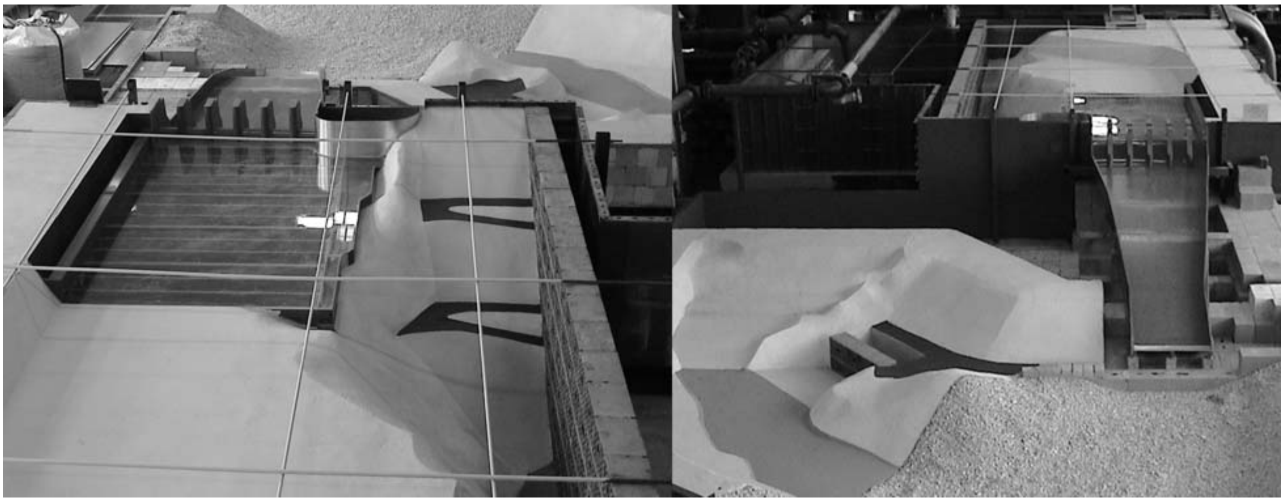

Figure 11.- Scale model at scale 1/100 – View of the upstream basin (left) and general view (right)

CONCLUSIONS

In this paper, a suitable scale model basin geometry has been defined for the study of a large dam thanks to previous numerical simulations of the general flow patterns in the reservoir. This reduced layout of the model basin has been validated regarding detailed numerical simulations results. The free surface flow numerical simulations have been realized using the flow solvers of WOLF software package.

Compared to the initial suggested layout, a gain of up to 30 percents in the upstream basin surface has been achieved. Simpler feeding conditions are also needed, and thus the delay and costs for the model building could be reduced.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge EDF-CIH, particularly MM. P. Gaudron and P. Marietta, for provided data and for the fruitful collaboration.

REFERENCES

Archambeau, P., Erpicum, S., Mouzelard, T., Pirotton, M. (2001). ”Impact studies and water management with WOLFHYDRO : a new physically based hydrological solver”. In *Proc. Int. Symposium on Environmental Hydraulics*. Arizona State University, USA.

Dewals, B., Archambeau P., Erpicum, S., Mouzelard T., Pirotton M. (2002). ”Coupled computations of highly erosive flows with WOLF software”. In *Proc. 5th International Conference on Hydro-Science & -Engineering*, Warsaw University of Technology, Poland.

Dewals, B., André, S., Schleiss, A., Pirotton, M. (2004). ”Validation of a quasi-2D model for aerated flows over mild and steep stepped spillways”. In *Proc. of the 6th inter. conf. on Hydroinformatics*, Singapore.

Dewals, B., Erpicum, S., Archambeau, P., Detrembleur, S., Pirotton, M. (2006). ”Depth-integrated flow modelling taking into account bottom curvature”. *Journal of Hydraulic Research,* IAHR. In press.

Erpicum, S., Dewals, B., Archambeau, P., Detrembleur, S., Pirotton, P. (2004a). ”Global design of hydraulic structures optimised with physically based flow solvers on multiblock structured grids”. In *Proc. 9th Int. Symposium on River Sedimentation*. Yichang, China.

Erpicum, S., Dewals, B., Archambeau, P., Detrembleur, S., Fraikin, C., Pirotton, P. (2004b). ”Scale modelling and similarity laws for the study of an under pressure settling structure”. In *Proc. 9th Int. Symposium on River Sedimentation*. Yichang, China.

Erpicum, S., Archambeau, P., Dewals, B., Detrembleur, S., Pirotton, P. (2006). ”Fluid - structure interaction modelling with a coupled 1D - 2D free surface flow solver”. In *Proc. of Riverflow 06*. Lisbon, Portugal.

Pirotton, M. (1994). *Modélisation des discontinuités en écoulement instationnaire à surface libre. Du ruissellement hydrologique en fine lame à la propagation d'ondes consécutives aux ruptures de barrages*, PhD Thesis, University of Liege.

Pirotton, P., Lejeune, A., Archambeau, P., Erpicum, S., Dewals, B. (2003). “Numerical-experimental interaction in hydrodynamics: an integrated approach for the optimal management of hydraulic structures and hydrographic basins”. In *Proc. of the* 9^{th} *Int. Conf. on Enhancement and Promotion of Computational Methods in Engineering and Science*, University of Macau, Macao.