Numerical – experimental interaction in hydrodynamics: an integrated approach for the optimal management of hydraulic structures and hydrographic basins

M. Pirotton, A. Lejeune, P. Archambeau, S. Erpicum & B. Dewals
Laboratories of Fluid Mechanics, Applied Hydrodynamics and Hydraulic Constructions, University of Liege, Belgium
Fifth author: Research Fellow of the National Fund for Scientific Research (Belgium)

ABSTRACT: The present paper presents studies and tools devoted to the reduction of uncertainties in the scientific knowledge of various unsteady flows, including solid transport effects, in the perspective of applications such as restoration, rehabilitation and enhancement of riverbeds, rivers and waterways. This global approach will be tested and fitted in a dynamic and integrated experimental model of reference flows in the laboratory. The relevance of using of physical experimental scale models will be highlighted for the identification of constitutive laws used in numerical modelling.

Practical applications prove that the integration of powerful hydrodynamic software for water management is realistic and will lead to compute transient flooding, sedimentation and erosion events in natural compound channels, as well as to design and simulate the behaviour of hydraulic regulation works or locks. It will also help in working out new projects or reconstructions of hydraulic infrastructures, as well as ecological rehabilitations. Even if the sole quantitative plan will be taken into account, this optimisation project will also lead to highlight favourable solutions, from a qualitative point of view.

1 INTRODUCTION

Practitioners in hydro-engineering are highly interested in the development of powerful tools for the prediction of a large scope of hydraulic behaviours. Scale models have been considered for a long time as an efficient approach for the design of hydraulic structures. Provided the right similitude law is used, they give access to reliable and uncontested results. Such experimental tests on scale models remain attractive for highly complex situations while hydrodynamic models fail to predict with confidence complex effects such as turbulence.

However, physical modeling suffers from several intrinsic defaults: high cost for building and measurement equipments, low geometrical flexibility once the model has been constructed, etc. In order to circumvent these problems, physical experimentations are more and more advantageously completed by numerical simulations. In this way, the quasi-three-dimensional flow solver WOLF is currently used at the University of Liege in interaction with physical models.

This approach leads to a suitable complementary work both at the stages of designing the scale model and during its exploitation. Typical applications include various aspects of hydrographic basins sustainable management and hydraulic structures design in the scope of the global change. The aim of this paper is to illustrate the benefits gained with the combination of both physical and numerical modeling.

2 FREE-SURFACE FLOW SOFTWARE WOLF 2D

WOLF 2D is an efficient analysis and optimization tool, which has been completely developed for several years in the Service of Applied Hydrodynamics and Hydraulic Constructions (HACH) at the University of Liege (http://www.ulg.ac.be/hach). WOLF 2D is part of WOLF free surface flows computation package, which includes in the same development environment the resolutions of the 1D and 2D depth-integrated Navier-Stokes equations as well as a physically based hydrological model, along with powerful optimization capabilities based on Genetic Algorithms (WOLF AG). See Figure 1 for the general organization of WOLF computation units. Each code handles structured or unstructured grids, dealing with natural topography and mobile bed simultaneously, for any unsteady situation with mixed regimes (including moving hydraulic jumps).

WOLF 2D software solves the 2D shallow-water equations on multi-block grids, dealing with natural topography and mobile bed. WOLF 2D is part of WOLF free surface flows computation package, which has been completely developed at the University of Liege. A mass balance for bed load sediments is coupled to the hydrodynamic model (see Dewals & al. 2002a or Dewals & al. 2002b). Side slope stability analysis are systematically performed to take gravity induced solid discharges into account.

The same finite volume technique in each computer code solves the equations, formulated in a conservative form to ensure exact mass and momentum balance, even across moving hydraulic jumps. An original splitting of the convective terms has been specifically developed for the model, in order to handle properly transient discontinuities. The computation core has reached now a high degree of reliability. Its stability, robustness and accuracy have been widely highlighted. Indeed the validation of the model has been performed continuously by comparisons with analytical solutions, field and laboratory measurements available in the literature or collected in the Hydraulic Laboratories in Liege.

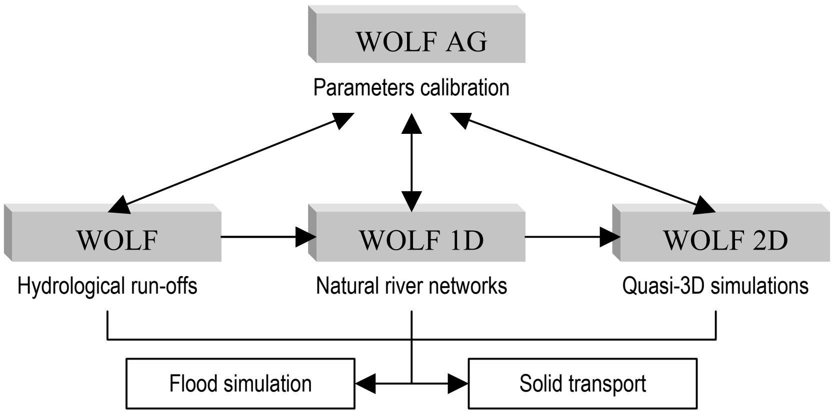

Figure 1: general organisation of WOLF computation units.

2.1 *Mathematical model*

The governing equations for hydrodynamic free surface flows are the depth-integrated Navier-Stokes equations for an incompressible fluid:

$$\frac{\partial h}{\partial t}+\frac{\partial h u_j}{\partial x_j}=0 \tag{1}$$

$$\frac{\partial u_i}{\partial t}+u_i\frac{\partial u_j}{\partial x_j}+g\frac{\partial H}{\partial x_i}\cos\theta_i+g\,n^2\frac{\sqrt{u_j^2}}{h^{4/3}}u_i=g\sin\theta_i+\frac{\partial}{\partial x_j}\left(\nu_t\frac{\partial u_i}{\partial x_j}\right)+G_i \tag{2}$$

where the Einstein notation has been used (sum over repeated subscripts). This system expresses the mass balance and the momentum balance along both space directions. The following symbols have been used: g (gravity acceleration), h (water height), n (Manning roughness coefficient), u_i (velocity components), t (time), x_i (space coordinates), θ_i (bed slope), G_i (external forces), ρ (water density) and ν_t (turbulent viscosity).

Equation (1) expresses the mass conservation, and equation (2) the momentum conservation along the x- and y-axis.

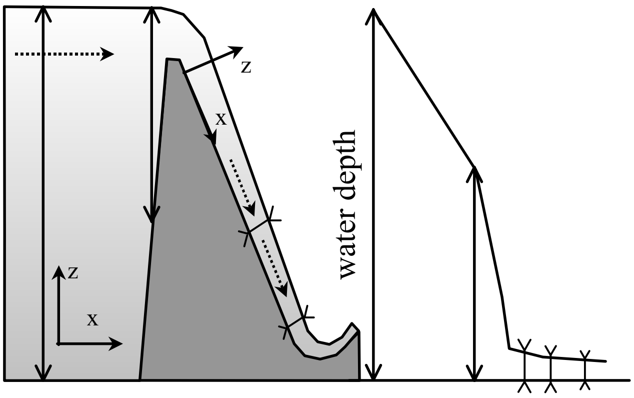

Figure 2: Axis inclination.

The total kinematic viscosity is given by the following expression:

$$\nu_{\mathrm{t}} = \nu + \gamma \sqrt{2\left(\frac{\partial u_j}{\partial x_j}\right)^2 + \left(\frac{\partial u_1}{\partial x_2} + \frac{\partial u_2}{\partial x_1}\right)^2} \ . \tag{3}$$

It must be outlined that no restrictive assumption is required for the bottom slope (see Pirotton 1997 or Mouzelard 2002). In order to simulate flows on very steep topographies (such as spillways or torrents for example), local references are defined with the x- and y-axis following locally the mean bottom slope (see Figure 2). The effect of the local bed curvature is taken into account thanks to a curvilinear system of coordinates. This technique ensures the water depth to be orthogonal to the local main flow direction.

According to the well-known Exner equation, the mass balance for sediments can be stated as

$$(1-p)\frac{\partial z_{\mathrm{b}}}{\partial t} + \frac{\partial q_{\mathrm{b}x}}{\partial x} + \frac{\partial q_{\mathrm{b}y}}{\partial y} = 0 \tag{4}$$

where z_{b} stands for the bed level, p is the porosity and $q_{\mathrm{b}x}$, $q_{\mathrm{b}y}$ represent the solid discharges in both horizontal directions. The bed load discharge is evaluated for instance using the Meyer-Peter and Müller formula, known to be widely reliable:

$$q_{\mathrm{b}} = 8\sqrt{(s-1)\,g\,d^3}\left[\xi\frac{R_{\mathrm{h}}\,J}{(s\text{-}1)\,d} - 0.047\right]^{\frac{3}{2}} \tag{5}$$

s represents the relative density of the sediment particles, d is the mean grain diameter, J stands for the energy slope and R_{h} for the hydraulic radius. Several other solid discharge laws are available within the computation program and the user is free to choose any of them.

2.2 *Space discretisation*

The finite volume method is used to achieve the space discretization. This approach is recognized to be especially adequate for highly advective flow conditions because it allows an easy implementation of upwind schemes. Moreover the numerical treatment by finite volume ensures exact mass and momentum conservation even across moving discontinuities. These two properties are of first interest for computing dam-break induced flows.

A remaining challenge lies in the design of a stable and efficient flux evaluation scheme. The stability requirements imply that whatever the flow regime no numerical oscillation may be generated, especially in the vicinity of discontinuous solutions (e.g. hydraulic jumps or sediment bores). Roe's approximate Riemann solver as well as an original Flux Vector Splitting technique are implemented in WOLF's numerical model. Both methods showed their ability to simulate sharp transitions without excessive smearing. Our original scheme is based on an upstream evaluation of purely advective terms (Dewals & al. 2002a, Mouzelard 2002), hence it takes into account the basic physical meaning of these transport terms in the mathematical model. The scheme, first developed for pure hydrodynamics, has been extended to the fully coupled system including the Exner equation. One key advantage is that the flux evaluation in the coupled model remains in accordance with the numerical scheme developed so far for the clear-water hydrodynamics. The numerical scheme has also proven its ability to reproduce automatically the phenomena of dunes and antidunes propagation observed in the nature (depending on the flow regime). The scheme is Froude independent, so that the critical transition doesn't call for any particular treatment

2.3 *Time discretisation*

As we are mostly interested in transient flows, an accurate and non-dissipative temporal scheme has to be chosen. The explicit Runge-Kutta schemes are used in WOLF both for their relatively low computation cost in the case of transient phenomena and for the high order of precision they allow to reach.

Implicit time integration is now available within WOLF software. This technique constitutes a brilliant strategy for computing highly accurate steady state solutions as it dramatically cuts the computation time for such a case. However due to the large algebraic systems to be solved at each time step implicit methods remain inappropriate for very unsteady flow conditions, where small time steps have to be used to assure a sufficient time accuracy and to track reliably each wave movement

3 THE NAM THEUN LARGE DAM SCALE MODEL

The Nam Theun II dam is a part of a set of three dams located on the Nam Theun river in Laos. A gated spillway is integrated in the main structure, and is followed downstream by a stilling basin. The Laboratories of the University of Liege have undertaken the study of the hydraulic scale model of the spillway and the stilling basin of the Nam Theun II dam.

The topography of the upstream basin shows that the dam is located near an important meander, which should *a priori* be included in the geometry of the scale model to ensure a correct stream orientation at the approach of the spillway.

However, the need of high precision measurements on the spillway induced the physical model to be scaled at 1 to 60. Therefore, the geometry of the scale model was restricted to the last part of the upstream basin (see the dotted line on Figure 3), the spillway with its stilling basin, and the first part of the downstream river (see Figure 3).

In order to respect the real upstream flow orientation and repartition, some numerical simulations were decided to be performed to delineate the upstream scale model boundaries.

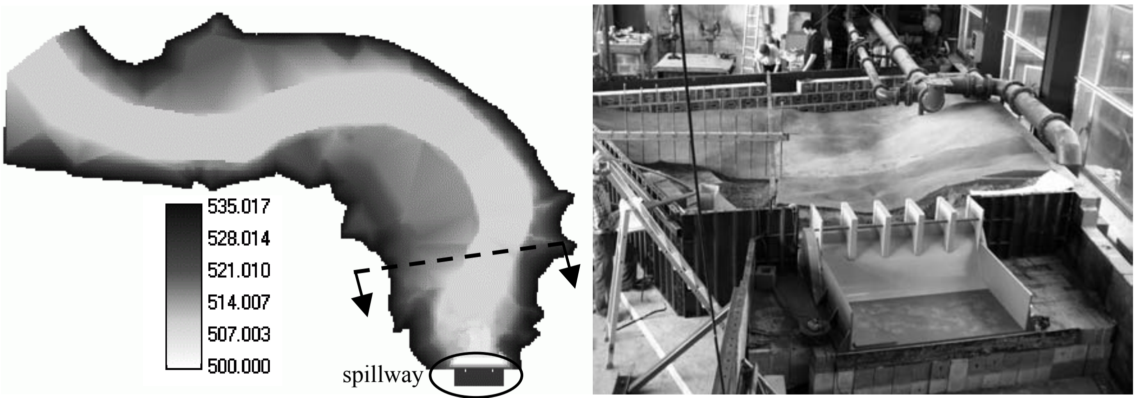

Figure 3: Topography (m) of the upstream basin and general view of the scale model.

3.1 *Upstream part*

The building of a scale model including a large upstream area was impossible due to the scale used and the space limitation. A first series of numerical simulations covering the whole reservoir flow (Figure 3) of the dam was performed in order to compute the flow field near the spillway without influence of the upstream boundary conditions.

The maximum discharge was numerically introduced by an upstream filling basin. No downstream boundary condition was needed (supercritical flow), thanks to the use of properly inclined axes, as seen in paragraph 2.1.

The results computed show that the main stream leaves the river path, and flows in the floodplain. This causes a highly asymmetric flow near the spillway. This latter is indeed found to be more effective through its right side (considered in the stream direction), rather than through the left one as it could be expected by the sole topography observation (see Figure 3).

3.2 *Geometry of the scale model*

A second set of simulations was performed based on the geometry selected for the scale model. The results strongly differ from those obtained on the global geometry, and confirm the poor analogy of the discharge distribution at the upstream part of the scaled geometry, see Figure 4. It is obvious that the main flow follows the river bed path, in absence of the upstream bend.

To improve the behavior of this reduced geometry in comparison with the whole reservoir and to ensure correct upstream flow conditions for the scale model, several modifications were decided for the upstream part of the reservoir.

The first modification concerns the topography. In concordance with the scale model, the computation simulates an upstream filling basin. This means that there is no upstream discharge or speed boundary condition, but only a solid wall. The discharge is introduced in the geometry by a bottom negative infiltration process, which avoids any arbitrarily imposition of velocity directions, and ensures a perfect fitting with the scale model conditions.

As the filling procedure depends on the topography, it was decided to modify this latter in order to get the same discharge distribution as in the global, previous simulation. The river bed of the filling basin was thus fundamentally modified (Figure 4).

The second modification concerns the adjunction of deflectors to the upstream basin. The deflectors are shown on the Figure 5. Several simulations were performed to determine their number and to estimate their optimal length and orientation in the scope of minimizing the difference with the large scale simulation. The bed modification ensures a good upstream discharge distribution, while the deflectors forces the flow into the right orientation.

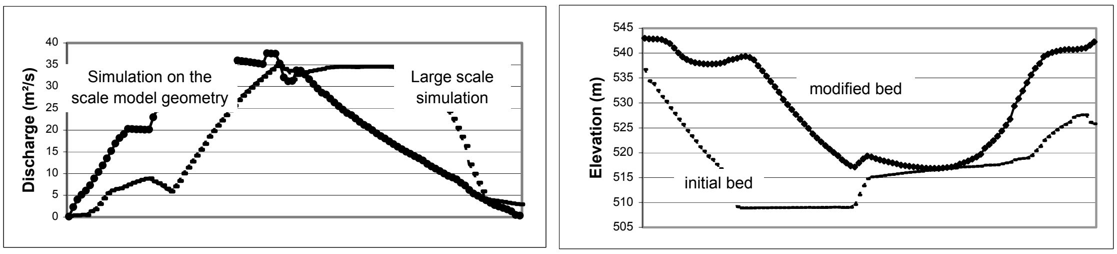

Figure 4: Discharge repartition along the transversal section corresponding at the upstream limit of the scale model geometry (left) and bed elevation of the scale model stilling basin (right).

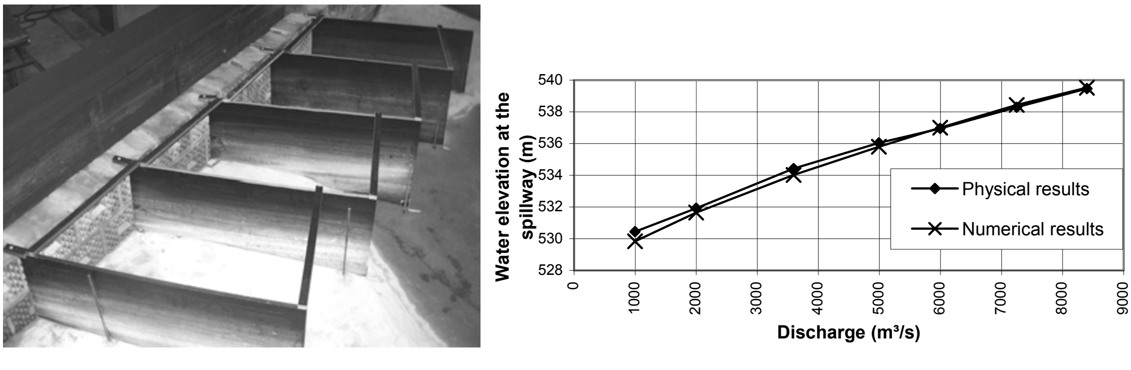

Figure 5: Deflectors of the scale model (left) and discharge rating curve of the spillway (right).

The different measurements on the scale model enable a comparison between physical and numerical simulations. Figure 5 proves the good agreement obtained in the estimation of the height/discharge curve related to the spillway.

3.3 *Stilling basin and river restitution*

The dam spillway is completed by five large gates to regulate accurately the discharge and the upstream level. However, the operations with asymmetric gate opening focused the attention because of the erosion risk in the downstream natural river. A complete numerical and physical hydrodynamic study of the stilling basin design was thus performed to induce and control dissipation for a wide range of discharges and gate opening configurations.

Conclusions obtained from physical tests and numerical simulations showed that some opening configurations should be avoided to keep erosion process at an acceptable level. For example, Figure 6 shows the velocity field in the situation of a low discharge flowing through the right gate only. Such situation produces an extreme erosion downstream of the stilling basin and must be rejected.

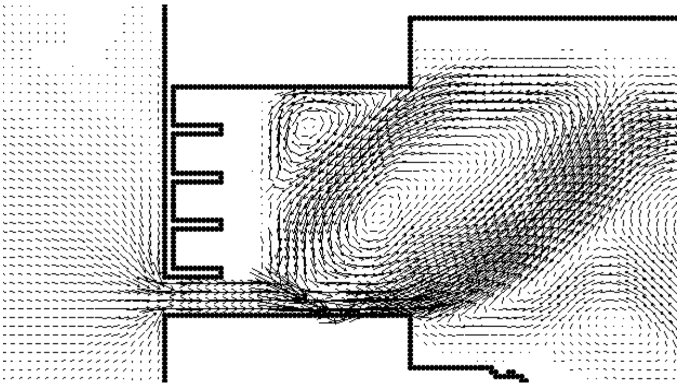

Figure 6: Velocity field in the case of a single gate opened.

4 THE KOL DAM SERIES OF NUMERICAL AND SCALE MODELS

The Kol Dam project includes the building of a very large embankment dam, with a crest 500 meters long at the level 648 m, as well as a complex of desilting structures. The exploitation level will vary between 642 and 636 m. The spillway is made up of 6 bays 17.1 meters wide, separated by 6 meters wide piers. The opening of the bays is regulated by sector gates. The spillway has a length of 420 meters, a variable width of 108.5 meters upstream to 70 meters downstream, and is ended by a flip bucket. An approach channel 230 m wide is located upstream of the spillway and of the desilting structure.

This structure is made of 14 identical desilting chambers placed side by side, perpendicularly to the spillway axis, upstream of it. These huge chambers are 16.42 meters wide, 12 meters high and 180 meters long. Their bottom is made of pyramidal shaped hoppers

4.1 *General strategy*

Any hydraulic project of such a high level of complexity involves both critical hydrodynamical considerations and solid transport aspects. Moreover those two types of interacting features include intricate processes such as fully threedimensional turbulent flows (water intakes), possibly laden with suspended load (desilting structures, upstream reservoir), and even aerated flows (spillway, flip-bucket). It is thus clear that no numerical model is currently available to perform a reliable analysis of the design as a whole.

On the other hand, a design methodology based exclusively on physical modeling may not be seen as the most attractive approach. Indeed, the spatial extension of the dam with its upstream reservoir, as well as the crucial need to reproduce experimentally essentially unspoiled physical phenomena forces the scale of the physical model to remain large enough. Besides the faithful scaling of complex physical processes (such as sediment load capacity, turbulence spectrum or air entrainment), a preserved accuracy of the various measurements to be performed also forces the scale model to remain particularly large. Fulfilling those two requirements would force the modeler to undertake the building of an extraordinary big and costly scale model, especially at the stage of the design, when the final shape and geometry of the structure are not yet known.

As a consequence, the global project has been studied at the University of Liege thanks to a whole set of combined numerical and physical models. Both shades of modeling are carried out in continuingly interacting and complementary perspective. For each part of the whole structure to be designed, the modeling method bringing most advantages for the specific case has been selected: numerical, physical or a combination of both.

4.2 *Hydrodynamic processes*

Physical scale models have been used to perform a complete analysis of the following items:

- water intakes of the diversion channels, flow through them and downstream restitution
- water intake for the hydroelectric power plant
- approach conditions of the spillway.

Indeed those flow fields are mainly driven by turbulence as well as three-dimensional free surface flow conditions and are thus hardly impossible to handle reliably numerically at a reasonable cost. Figure 7 shows the global model and a more detailed view of the spillway.

Numerical modelling has been used simultaneously for instance to optimise the design of the water inflow into the upstream part of the global model, in such a way that effects related to the upstream reservoir topography, although not reproduced experimentally, remain properly reproduced. Figure 8 shows three-dimensional views of the Digital Elevation Model used in the numerical simulations.

Figure 7: global scale model (left) and detailed study of the scouring intensity induced by the impinging jet downstream of the flip-bucket (right).

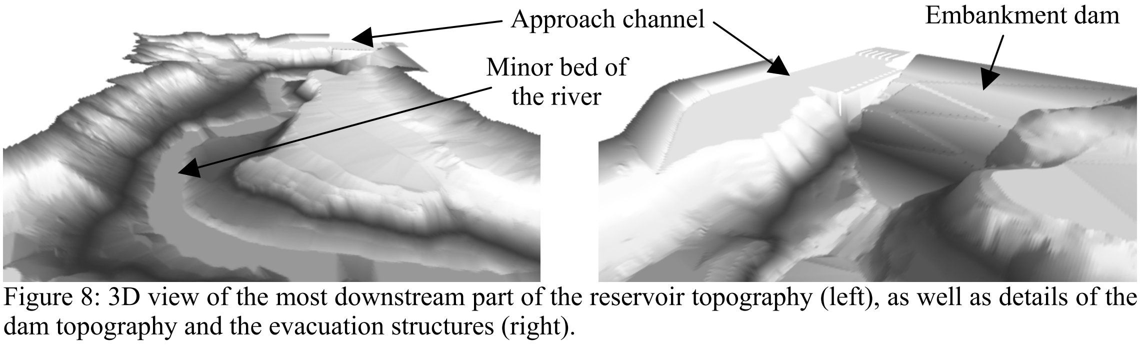

Figure 8: 3D view of the most downstream part of the reservoir topography (left), as well as details of the dam topography and the evacuation structures (right).

4.3 *Solid transport features*

Solid transport aspects of the project involve the design of the desilting chambers and their hoppers as stated above. The trap efficiency of those structures is mainly influenced by the turbulence spectrum of the sediment-laden flow crossing the chambers. Once more no available numerical model can perform such a simulation at an acceptable level of accuracy and cost. Two large scale models have thus been exploited.

On the other hand, the long-term sedimentation time in the whole reservoir can only be estimated thanks to a numerical approach because of the huge space- and time-scales of the process. This has been performed with WOLF in a quasi-steady approach (Figure 9). The *transient* aggradation and bed level evolution have been computed on the basis of successive steady-state hydrodynamic simulations.

Finally, the rapidly transient flow with highly erosive power during a flushing operation have been analyzed with the fully-coupled and unsteady numerical tools available in WOLF. The mains goals of this numerical study consists in predicting the effect of a given flushing scenario in terms of changes in bathymetry in the downstream area of the reservoir of Kol. The efficiency of the flushing operation has been be evaluated with respect to the recovered storage capacity, its extension in space and the location of the channel generated by the flushing.

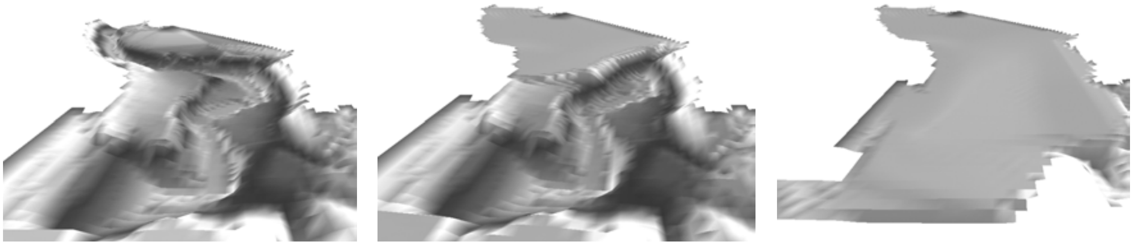

Figure 9: Bottom topography in the downstream part of the reservoir initially, after 12 years and after 24 years of silting.

5 CONCLUSION

An integrated approach for global studies on scale and numerical models was described. It demonstrates that the increase of the computers performance, and the efficiency of the actual free-surface flow solvers lead to useful numerical information for optimal scale model design and studies.

In this way, the software package WOLF developed by the Service of Applied Hydrodynamics and Hydraulic Constructions at the University of Liege is described as an efficient tool in the scope of the study and design of hydraulic structures or, more generally, of river flow management. The detailed examples of two large dams in Asia showed that the numerical simulations have allowed to reach precious time savings by circumscribing the scale model in the minimum part where a physical model is required. The numerical approach also brought valuable guidelines for the analysis of aspects of the project impossible to handle in a laboratory experiment.

REFERENCES

Dewals, B., P. Archambeau, S. Erpicum, T. Mouzelard and M. Pirotton (2002a). Dam-Break Hazard Mitigation with Geomorphic Flow Computation, Using Wolf 2D Hydrodynamic Software. International Conference on Risk Analysis, Sintra, Portugal, WIT Press.

Dewals, B., P. Archambeau, S. Erpicum, T. Mouzelard and M. Pirotton (2002b). An Intergated Approach for Modelling Gradual Dam Failures and Downstream Wave Propagation. First IMPACT Workshop, Wallingford (UK), HR Wallingford.

Pirotton, M. (1997). "Une Approche Globale pour Modéliser la Genèse et la Propagation des Crues Naturelles ou Accidentelles ", Classe des Sciences, Académie Royale de Belgique, 182 pages.

Mouzelard, T. (2002). Contribution à la Modélisation des Écoulements Quasi Tridimensionnels Instationnaires à Surface Libre. PhD thesis, University of Liege.

Roe, P.L., (1981). "Approximate Riemann solvers, parameter vectors and difference schemes", Journal of Computational Physics.

Venkatakrishnan, V., (1993). "On the accuracy of limiters and convergence to steady state solutions", AIAA paper.

Rogiest, P. (1997). An implicite finite volume scheme for the computation of unsteady compressible flows on multi-block structured grids. PhD thesis, University of Liege.