3D CFD modelling of spillways: Practical feedback on capabilities and challenges

Y. Oukid, V. Libaud and C. Daux, Tractebel Engineering (Coyne et Bellier), France

With the continuous development of computing capabilities, the numerical modelling of complex hydraulic flows has become more and more efficient and cost effective, providing a powerful tool for hydraulic analysis, from preliminary to detailed studies. This article describes how the 3D CFD tool has been first validated on a large range of hydraulic phenomena, then used at several large projects to analyse and optimize the design or operation of the hydraulic structures. Emphasis is on the challenges that still have to be overcome, in particular the impact of air entrainment.

Until recently, detailed studies of hydraulic structures were done almost systematically on physical hydraulic models, which have offered great capabilities in understanding the hydraulic behaviour of the flows studied. But physical models are generally rather expensive and time consuming, and they can only represent some of the hydraulic phenomena of interest, depending on the similarity law for which they have been calibrated. Nowadays, given the increasing computational capabilities and further development of user-friendly interfaces, 3D CFD modelling can balance these issues and provide a flexible and powerful tool to support physical modelling, or even replace it in certain cases.

In the first section, this article presents a selection of simulation cases carried out to validate the use of the Flow 3D© software and identify its limits. The performance of 3D CFD modelling, as well as some pending challenges, are then discussed, with a presentation of a selection of large hydraulic projects. The focus is on specific phenomena, such as high velocity flows, air entrainment, energy dissipation, and pressure fluctuations.

1. Validation cases

Flow 3D© is a multi-physics computational fluid dynamics (CFD) software package, developed by Flow Science Inc. The software approaches numerical solution of the Navier-Stokes differential equations, applying a finite-difference method with simple free rectangular grids, and a 'volume of fluid' (VOF) technique to track free surfaces.

Before applying Flow 3D© at projects, the software was thoroughly validated based on various specific cases to assess its capabilities, accuracies and validity range.

The validation cases were selected to cover the main hydraulic issues usually encountered in the field of dam spillways and appurtenant structures. It was further validated by comparison with references in the literature and experimental data from physical models.

The cases studied were, from upstream control and conveyance structures, to downstream energy dissipation structures:

- uniform flows (roughness influence and velocity profiles);
- flows in curves and bends;
- flows over broad-crested weirs, ogee crests and lateral spillways, under submerged and unsubmerged conditions (discharge capacity, flow profile and pressure profile);
- supercritical flows after a sharp expansion and behind a pier (stationary waves);
- flip buckets and ski jumps (flow profile, trajectories, bottom pressures and hysteresis);
- vertical jet in a plunge pool (velocity profile and bottom pressures);
- hydraulic jumps (flow profile, air entrainment and energy dissipation); and,
- stepped spillways (flow profile, air entrainment and energy dissipation).

Systematic sensitivity analyses were carried out on each of the predominant parameters for the various validation cases, making it possible to define a set of optimal parameters for each modelled phenomenon. Furthermore, quality criteria were defined, particularly with respect to the accuracy versus the relative mesh size and the computation time.

The following presents a brief summary of selected cases, from basic flows over weirs to hydraulic jumps, including air entrainment phenomena, as well as stationary waves formed in supercritical flows and ski jump hydraulics.

It is worth noting that, when convenient, the validation cases were tested using 2D vertical models. This method, involving only one cell in the transversal direction, makes it possible to implement preliminary models quickly. It is applicable to free-surface problems, where the transversal third dimension is not predominant. This type of 2D vertical model is very convenient and efficient for the engineer, as quick simulations and sensitivity tests can be done, along with determining the optimal configuration for a full 3D model, that is, if such a 3D analysis is required.

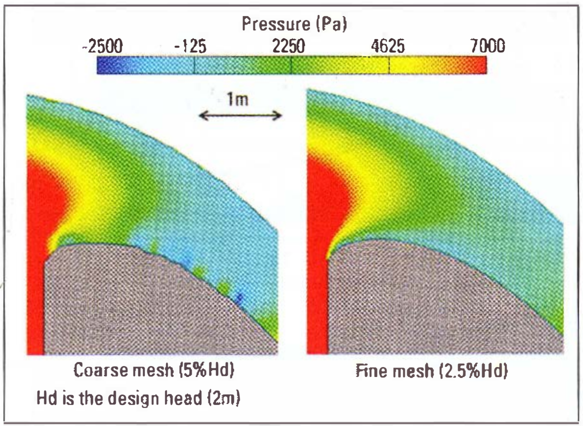


Fig. 1. Influence of mesh refinement on the pressure computation.

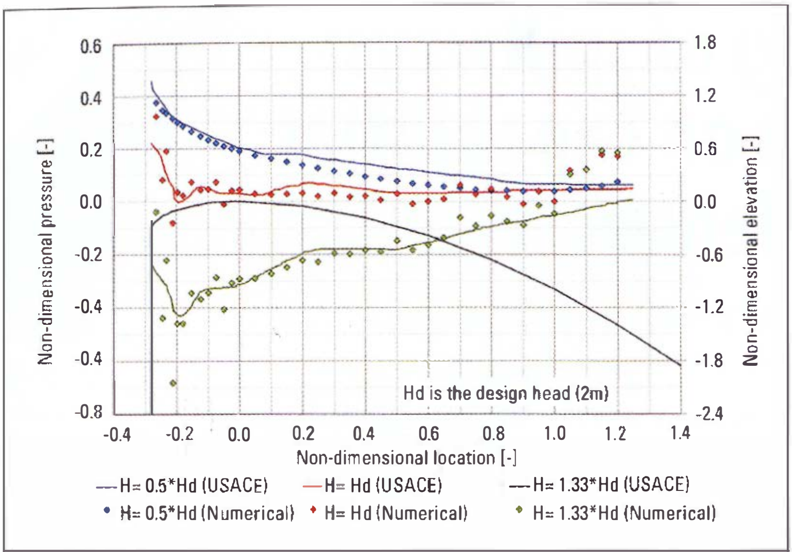


Fig. 2. Pressure distribution on the standard weir.

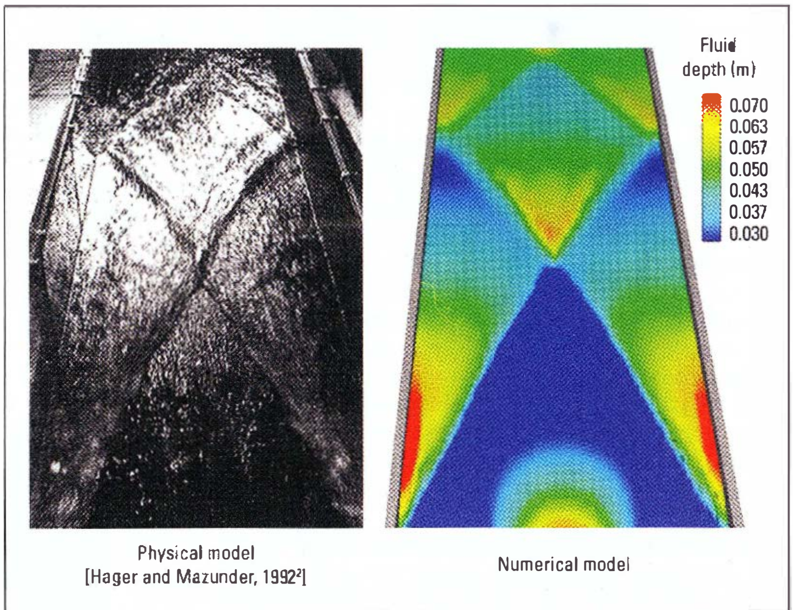


Fig. 3. Shock waves created by a sharp expansion under supercritical flow condition. Numerical and physical observation.

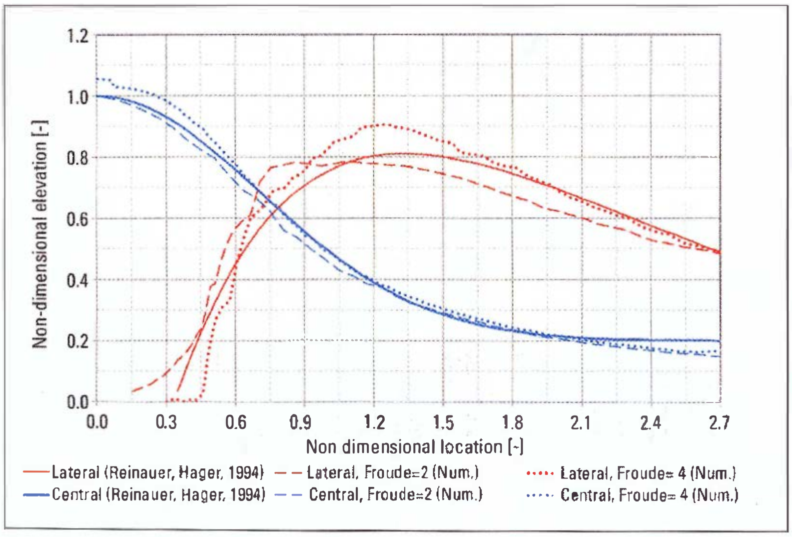


Fig. 4. Water profiles downstream of a sharp expansion: comparison of numerical results with experimental data.

1.1 Flow over weirs

The validation procedure was first applied to flows over broad-crested weirs and ogee weirs, for unsubmerged and submerged cases.

Not surprisingly, the accurate modelling of the hydraulic profiles, the discharge capacity and the pressure distribution is directly dependent on the mesh size.

The validation procedure concluded on an optimum mesh size, within the range of 5 to 10 per cent of the head on the weir, making it possible to achieve precision on the discharge ranging from 1 to 6 per cent, as per the literature reference [USACE, 1987[1]]. Accuracies within 0.2 per cent were also obtained with finer meshes, thus proving that the computation converges towards the expected results, but entailing rather long computation times. Hydraulic profiles and submergence coefficients were also reproduced with an adequate accuracy, under the same mesh conditions as for the discharge capacity.

Nevertheless, as shown in Fig. 1, the bottom pressure distribution is poorly represented with this set of parameters, since the mesh is too coarse to model the flow pattern correctly near the weir bottom.

Therefore, if the bottom pressure distribution is of interest for the modeller, local refinement near the solid interface should be adopted, through nested mesh blocks.

With this local refinement, the bottom pressure distribution can be estimated with a satisfactory accuracy, as shown in Fig. 2, which compares the numerical and experimental bottom pressure profiles for various heads.

1.2 Shock waves

To assess the capabilities of the software for modelling stationary shock waves that may occur, for instance in a spillway chute, the impact of a sharp expansion and a pier on supercritical flows have been simulated.

A full 3D numerical model was developed, requiring specific mesh refinements to reproduce the water elevations caused by shock waves accurately. The results were compared with experimental data from the literature [Hager and Mazunder, 1992[2]; Reinauer and Hager, 1994[3]] showing a very good correlation.

As for sharp expansions, shock waves with high water elevations developed from the lateral walls converging

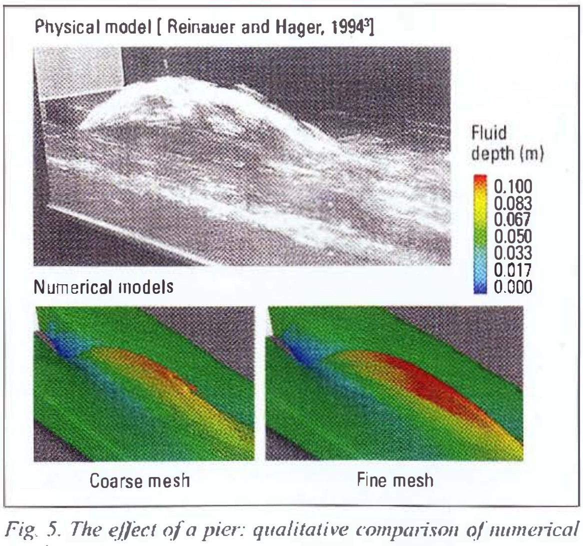


Fig. 5. The effect of a pier: qualitative comparison of numerical results with experimental observations.

towards the channel axis, and the numerical computations demonstrated that they were accurately represented, in terms of planar location and water elevations, see Figs. 3 and 4. An accuracy of less than 5 per cent was achieved for a wide range of Froude numbers.

The second case, corresponding to the study of piers effects on supercritical flows, was tested for various flow conditions. The results were also quite satisfactory. However, the small water thicknesses of the wave required significant local mesh refinements, see Fig. 5.

1.3 Ski jump hydraulics

Ski jumps are widely used in spillway structures, since they allow for the discharge of high energy flows away from the toe of dams.

Flip buckets and ski jump hydraulics have been modelled and tested for a large range of Froude numbers. 3D hydraulic modelling has been found to simulate very well water level profiles, jet trajectories and bottom pressure profiles with sufficient mesh refinement.

As an illustration, Fig. 6 compares the pressure profile on the flip bucket bottom obtained from 3D simulations with experimental data [Juon and Hager, 2000[4]; Heller *et al*, 2005[5]].

This shows that the 3D model results correspond well with the experimental results: the peak value and its position are reproduced properly, except for cases with Froude numbers higher than 10.

The operation of flip buckets at low flows has also been modelled, to check the capability of a 3D model to simulate the chocking flow conditions for which a hydraulic jump takes place, see Fig. 7, under increasing and decreasing discharge regimes.

It has been proved that the 3D simulation can model the hysteresis phenomena associated with such flow conditions appropriately. The computed increasing and decreasing discharge limits were in accordance with the experimental data, within an error margin of less than 3 per cent.

1.4 Hydraulic jumps and air entrainment

Hydraulic jumps are of great interest in dam engineering, since practical applications are frequent, especially at energy dissipation structures downstream of spillway structures.

A hydraulic jump involves a rapid variation in flow regime, from supercritical to subcritical, and generates turbulent rollers, allowing significant amounts of air to penetrate the flows. The flow occurring in a hydraulic jump is therefore biphasic.

The validation process focused initially on reproducing simple characteristics of hydraulic jumps, such as conjugated heights, energy dissipation and rollers lengths.

Satisfactory results were obtained for monophasic models, and the main hydraulic characteristics were reproduced with good accuracies compared with the literature [Rajaratnam, 1967[6]].

When aiming to reproduce the air entrainment process using biphasic models, the modelling became more complex.

A more complete calibration process, involving turbulence models, momentum advection, and various air entrainment modules, was therefore carried out. This was to assess the capabilities of the software to reproduce air entrainment in hydraulic jump, and to verify its impact on the flow characteristics and energy dissipation.

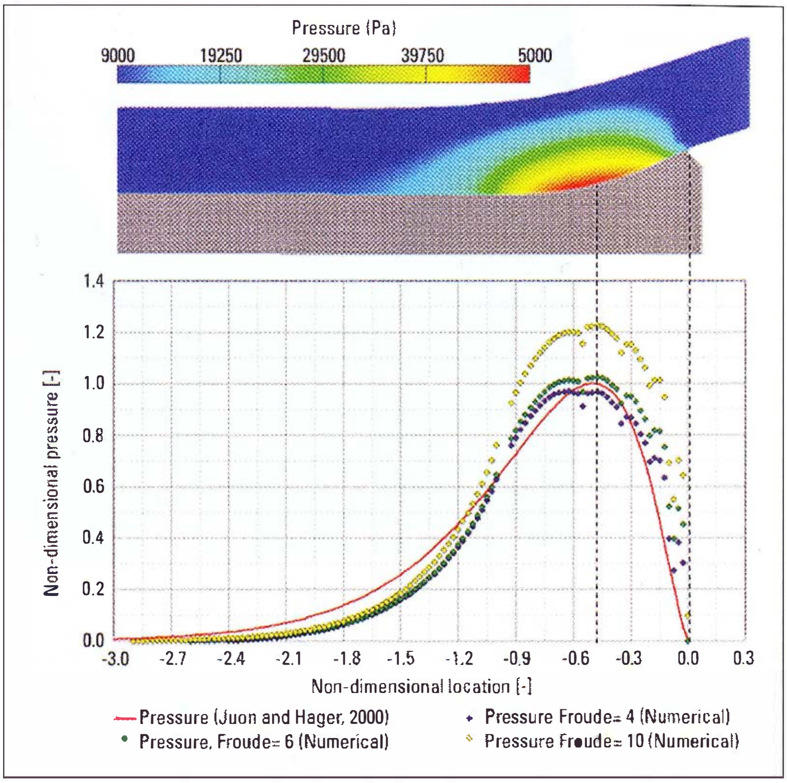


Fig. 6. Pressure head distribution on the flip bucket apron: comparison of numerical results with experimental data.

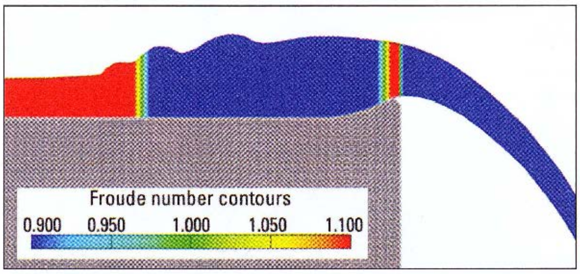


Fig. 7. Froude number contour on hydraulic jump formed in a flip bucket, in the case of increasing discharge.

The calibration tests made it possible to highlight the relationship between turbulence levels and air entrainment, which influence each other. It demonstrated that, even if for some model configurations the hydraulic profile was reproduced reasonably, turbulent levels and air entrainment can reveal strong discrepancies.

Fig. 8 illustrates, for instance, the influence of the turbulence model (RNG model versus k-ε model) on the air entrainment, as well as on the hydraulic jump position, the RNG model being the one to be clearly preferred.

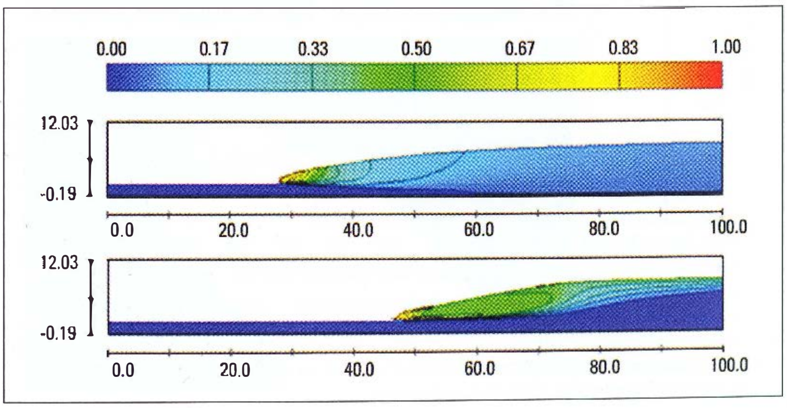
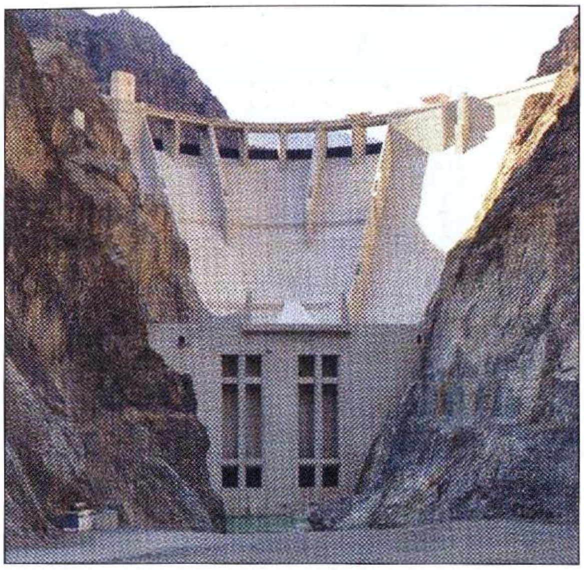


Fig. 8. Hydraulic jump-influence of turbulence model (Air entrainment volume ratio. Upper: RNG model, lower : k-ε model).

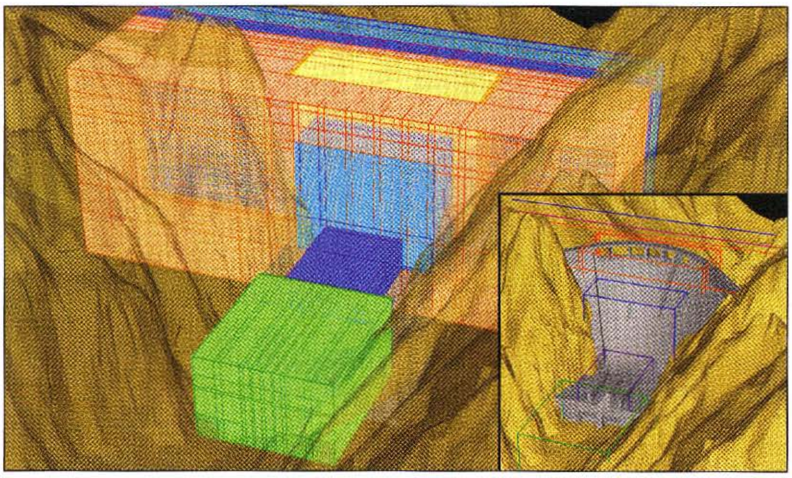
Downstream view of the Artvin spillway.



After this calibration phase, the biphasic model was able to reproduce hydraulic jump with a precision of between 5 and 10 per cent on the volume of entrained air, and between 10 and 20 per cent for the energy dissipation efficiency.

Nevertheless, the calibrated model parameters were strongly dependent on the type of hydraulic jump, that is, the Froude number.

Fig. 9. 3D numerical model of Artvin dam: the grid system.



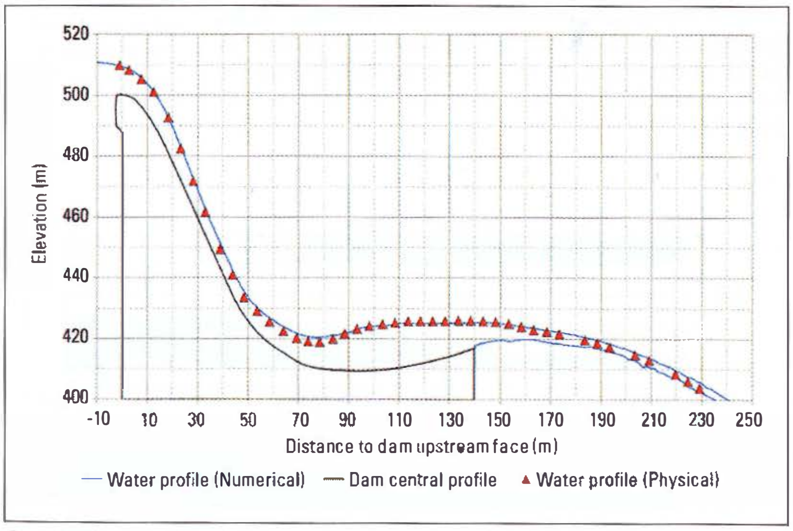


Fig. 10. Flow profile in longitudinal central section of the Artvin spillway: Comparison of numerical and physical results.

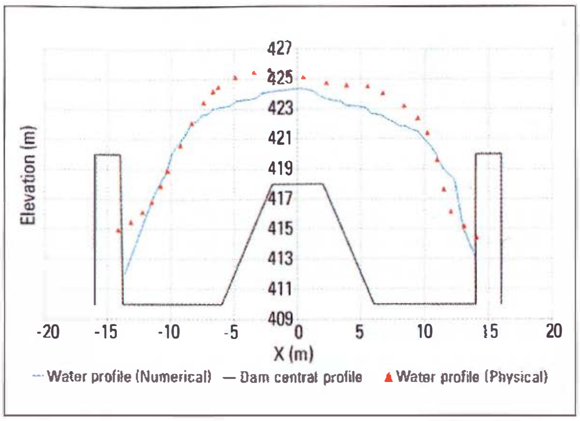


Fig. 11. Flow profile in the downstream transversal section of Artvin spillway: Comparison of numerical and physical results.

2. Project application examples

2.1 High velocity spillway flow

The Artvin hydro project is located on the Coruh river, in northeastern Turkey. The principal components of the project are a 180 m-high thick arch dam, and a 332 MW powerplant, see photograph.

The spillway is located on the arch dam body, the powerhouse roof supporting the downstream end of the spillway chute. The spillway crest is equipped with seven sluice gates, providing a discharge capacity of 8200 m^3/s. The spillway chute is convergent with two separating walls on the upper part and a central deflector.

The detailed design carried out in 1990 foresaw an ungated spillway. Nevertheless, gates were added at the beginning of the construction stage, to increase the total head, thus increasing the power potential.

The purpose of the 3D numerical model was to assess the hydraulic behaviour of the gated spillway flow under various operating conditions.

Because of the complex 3D geometry and the high velocities involved, special attention was paid to the mesh construction. The grid system developed for this model was made of 10 different blocks, three being nested, from a 4 m mesh size to 0.3 m. The result was a total raw number of 15 million cells. To optimize the

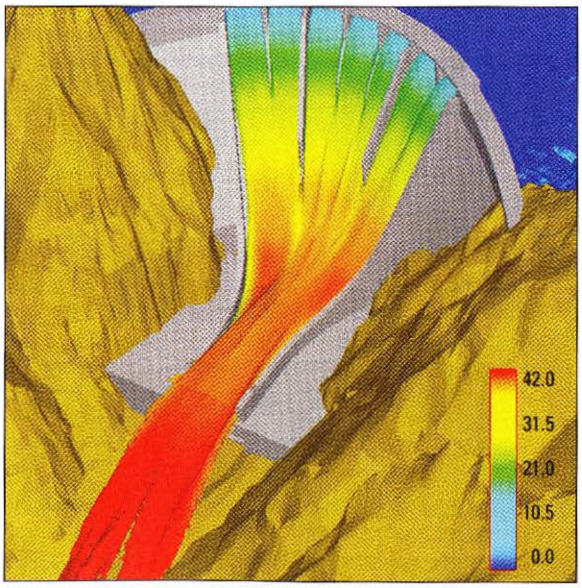


Fig. 12. 3D numerical results: Velocity field (m/s).

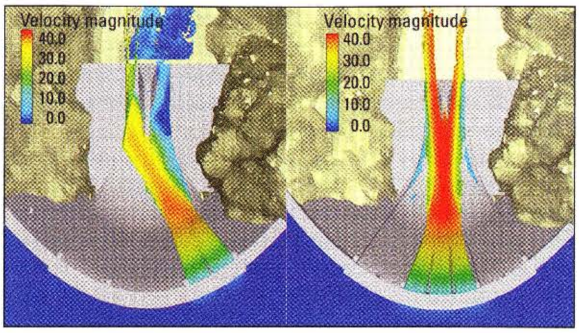


Fig. 13. Example of symmetrical and asymmetrical operating conditions.

computation time, the analysis of the preliminary results allowed for a reduction in the total number of active cells to 5.2 million, as the cells not directly involved in the flow patterns were deactivated.

A benefit of the physical modelling carried out during the final design stage in 1990, was that experimental results without gates were available and could be used to validate the 3D numerical model.

The spillway discharge capacity was reproduced within 5 per cent accuracy and the computed hydraulic profile has shown to be matching experimental results, as shown in Figs. 10, 11 and 12.

Various symmetrical and asymmetrical configurations of gates were then simulated to confirm the proposed operating rules of the now gated spillway The 3D numerical model was also used to calculate physical variables that were not available with the physical model, such as the pressure and the cavitation index on the whole spillway chute surface, making it possible to validate the design of the aeration system.

2.2 Hydrodynamics within a plunge pool

The Kariba dam is a concrete arch dam (128 m high with a crest length of 617 m), located in the Kariba Gorge of the Zambezi river between Zambia and Zimbabwe in Southern Africa.

Its spillway consists of six submerged sluices, equipped with Caterpillar gates of 8.8 m height and 9.15 m width with a total discharge capacity of about 9000 m^3/s under 30 m hydraulic load. As a result of sustained spilling periods, between 1961 and 1981, an exceptional plunge pool, 80 m depth, has been scoured in the central section of the riverbed, see Fig. 14.

2.2.1 Physical hydraulic modelling of the pool

In 2011, as part of detailed studies of the plunge pool reshaping, a hydraulic model (scale 1/65) was built by the Laboratory of Hydraulic Constructions of EPFL for the Zambezi River Authority (ZRA) [Bollaert *et al.*, 2012[7]; Noret *et al.* 2013[8]].

This model provided general flow characteristics and specific measurements of pressure and velocity of flow in the plunge pool. Pressure and velocity fluctuations were measured along longitudinal and transversal profiles for various scenarios of gate openings, using calibrated piezo-resistive transmitters and Acoustic Doppler Velocimetry (ADV).

The opportunity was seized to assess the capabilities of a 3D numerical model to reproduce the measured complex hydrodynamic phenomenon in this deep plunge pool, involving high velocities and turbulence levels, predominant air entrainment and pressure fluctuations.

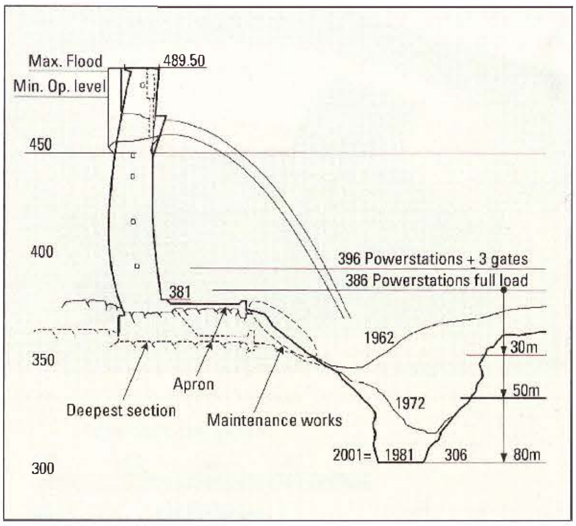


Fig. 14. Longitudinal section of the Kariba dam and its plunge pool.

2.2.2 Numerical approach

A 2D vertical model was first developed to perform sensitivity tests quickly on the mesh size, model configuration and boundary limits. The simulations showed that the specific shape of the spillway and the high velocities (up to 45 m/s) required particular attention for the spillway mesh definition, involving a multi-blocks approach. The model was validated by comparing the numerical results with the physical model measurements on discharge capacities and jet trajectories.

Based on the results of the 2D vertical model, a full 3D model was carried out to represent accurately the 3D hydrodynamic phenomenon observed in the Kariba plunge pool. This model involved about 1.5 active cells and significant calculation time compared with 2D approach.

The influence of air entrainment was analysed first, and then a specific analysis was carried out to assess the ability of the 3D CFD software to represent pressure fluctuations in the plunge pool.

2.2.3 Air entrainment

Air entrainment is a dominant phenomenon in flows emanating from high water falls. The high impact velocities generate major air entrainment and turbulence levels within plunge pools.

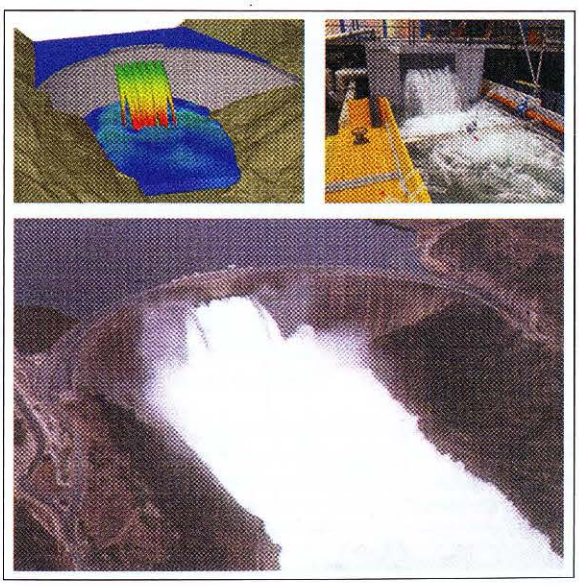


Fig. 15. Geometry of the 3D Kariba dam model and pool (upper left) together with EPFL's hydraulic model (upper right) and photograph of the commissioning test in 1966.

Fig. 16. Air entrainment with six gates fully opened at Kariba dam.

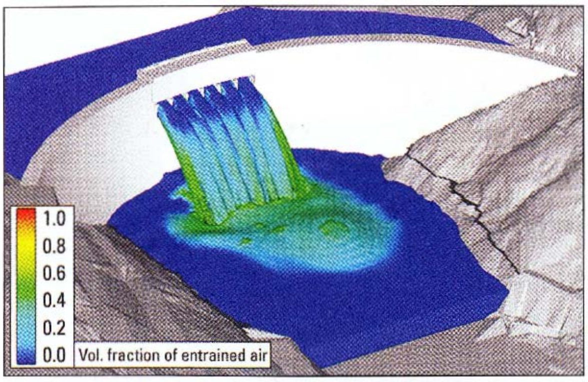


Fig. 17. Velocity distribution in a longitudinal section of the Kariba plunge pool.

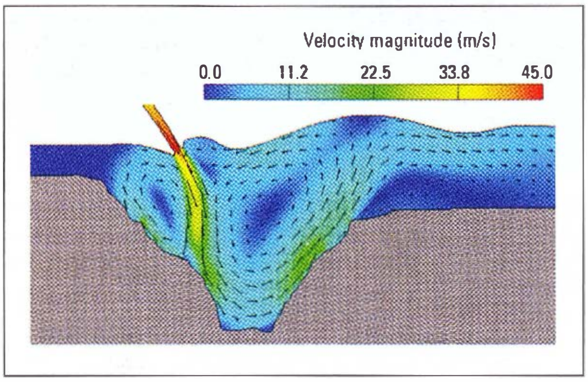
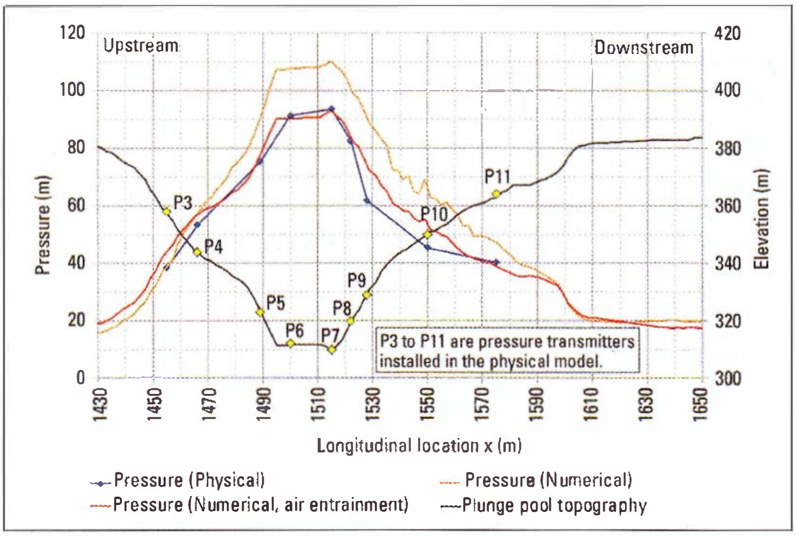


Fig. 18. Mean pressure at the bottom of Kariba plunge pool. Physical and numerical values.



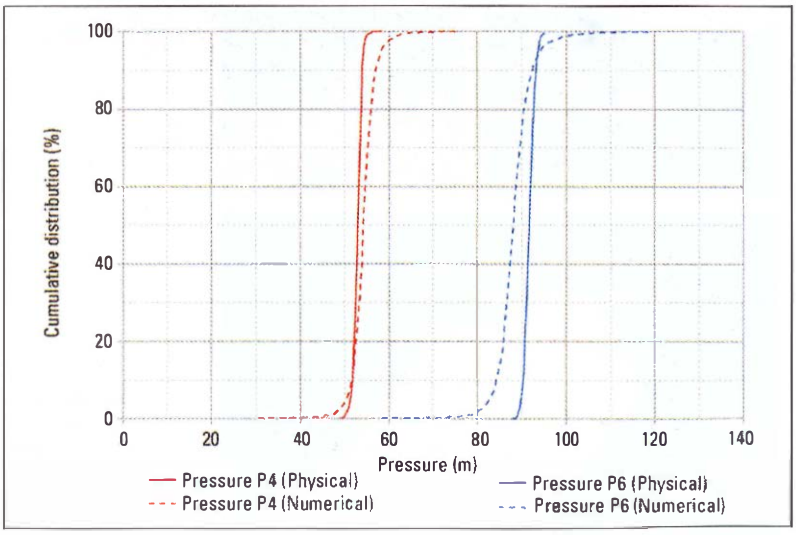


Fig. 19. Dynamic pressure at the bottom of Kariba plunge pool for pressure transmitters P4 and P6. Physical and numerical values.

The calibration of air entrainment parameters was shown to be very sensitive. In the absence of physical model data on entrained air, indirect calibration tests were done to compare the pressures and velocity distribution with measured data.

The numerical results showed that air entrainment has a significant influence on jet trajectories. Jet thicknesses in the air and impact velocities on the water surface are better simulated than with a monophasic model. The water density within the plunge pool is also significantly altered by the entrained air, impacting the pressure field, but also the velocity field within the plunge pool.

The numerical values of jet trajectories and velocities were compared both with the physical measurements and the empirical formula [Juon and Hager, 2000[4]], giving a precision within 5 per cent for the jet trajectory and the velocity at the impact with the plunge pool (instead of about 10 per cent without air entrainment model activated).

2.2.4 Flow recirculation

In addition, the velocity distribution in the plunge pool was compared with physical measurements. The recirculation in the downstream part of the plunge pool was demonstrated to be well reproduced. Moreover, it was observed on the physical model that the water jets were diverted under water, such that their incidence angles tended towards vertical, probably because of the strong downstream recirculation. This phenomenon was reproduced well by the numerical model, see Fig. 17.

2.2.5 Dynamic pressure

The comparisons of physical pressures recorded at the bottom of the plunge pool with the numerical values demonstrated that the mean pressure was simulated with sufficient accuracy. However, as for the velocity field, the results showed a relatively strong sensitivity to the air entrainment, requiring a specific calibration study. It could be observed that an increase of air entrainment induces a significant decrease of the mean pressure values, see Fig. 18.

The study also aimed to reproduce the dynamic pressure fluctuations. At most of the locations, it could be observed that the amplitude of dynamic pressure fluctuations was larger on the numerical model, see Fig. 19. Moreover, the interpretation of the pressure fluctuations proved to be challenging, since the hydrodynamic fluctuation frequencies are very high compared with the computation time step, which is limited by various model constraints as well as computational capabilities.

3. Conclusion

Physical phenomena involved in monophasic flows, however complex, have been proven to be well simulated by 3D CFD softwares. This requires that the modeller has sufficient hydraulic knowledge and experience to adapt a priori the model parameters, mesh and boundary conditions, and to carry out an *a posteriori* critical analysis of the results.

In spillway design, this statement applies in particular to the control and conveyance structures, such as channels, weirs, spillway chutes, or flip buckets.

When it comes to energy dissipation structures, such as stepped spillways, stilling basins or plunge pools, careful calibration tests of the air entrainment parameters must be carried out for each case, questioning the practical and stand-alone use of current 3D biphasic models in hydraulic engineering.

Nevertheless, once the air entrainment parameters have been calibrated, a hybrid approach for such complex analyses, combining physical and numerical models, is of great interest, as it can lead to time savings, a better physical understanding and enhanced flexibility in the design activities. ◊

References

1. **USACE**, Hydraulic Design Criteria, Elliptical Crest Spillway, 111-24 to 111-24/10; 1987.
2. **Hager, W.H. and Mazunder, S.,** "Supercritical flow at abrupt expansions". *Proceedings*, Institution of Civil Engineers Wat., Marit. & Energy, No 96(3), September 1992.
3. **Reinauer, R. and Hager, W.,** "Supercritical Flow behind Chute Piers." *Journal of Hydraulic Engineering, ASCE*, Vol. 120, No. 11, 1994.
4. **Juon, R., and Hager ,W.H.**, "Flip bucket without and with deflectors". *Journal of Hydraulic Engineering, ASCE*, Vol. 126, No. 11, 2000.
5. **Heller V., Hager W.H, and Minor, H.** "Ski Jump Hydraulics." *Journal of Hydraulic Engineering, ASCE*, Vol. 131, No. 5, 2005.
6. **Rajaratnam N.**, "Hydraulic Jumps", in: Advances in Hydroscience, New York and London, Academic Press; 1967.
7. **Bollaert, E.F.R, Munodawafa, M.C. and Mazvidza, D.Z.,** "Kariba Dam Plunge Pool Scour: quasi-3D Numerical Predictions", ICSE6 – 265, Paris, France; 2012.
8. **Noret, C., Girard, J-C., Munodawafa M.C, and Mazvidza D.Z.,** "Kariba dam on Zambezi river: stabilizing the natural plunge pool", *La Houille Blanche*, 2013.

Yanis Oukid is a Civil Engineer, specialized in hydraulic structures, graduated from Ecole Nationale Polytechnique d'Alger. He has been working on various design studies for large dams and hydroelectric power plants. He is currently working as project engineer on the 1200 MW hydroelectric dam project of Budhi Gandaki in Nepal.

Vincent Libaud is a Civil Engineer graduated from Grenoble INP ENSE3 in 2012, and specialized in hydraulic structures. He carried out Flow 3D validation tests and the numerical modelling of Kariba plunge pool. He currently works for the 3600 MW Rogun hydro scheme in Tajikistan.

Christophe Daux is graduated in Structural Engineering and in Water Resources Management. He is a member of the French Committee for Dams and Reservoirs and has been working on design studies and construction supervision for a number of large dams and hydroelectric power plants. He is currently deputy director for the Dams and Hydroelectric Schemes business unit of Tractebel Engineering.

Tractebel Engineering (Coyne et Bellier), Le Delage, 5 Rue du 19 Mars 1962, 92622 Gennevilliers Cedex, France.

Y. Oukid

V. Libaud

C. Daux