Utah State University

DigitalCommons@USU

International Symposium on Hydraulic
Structures

May 17th, 1:30 PM

Complementarities Between Physical Modelling and Computational Fluid Dynamics for an Ecological Continuity Project

M. Dufresne
3D EAU, matthieu.dufresne@3deau.fr

José Vazquez
ENGEES, jose.vazquez@engees.unistra.fr

Yvan Bercovitz
EDF, yvan.bercovitz@edf.fr

Follow this and additional works at: https://digitalcommons.usu.edu/ishs

Recommended Citation

Utah State University
MERRILL-CAZIER LIBRARY

Complementarities between physical modelling and Computational Fluid Dynamics for an ecological continuity project

M. Dufresne[1], J. Vazquez[2] & Y. Bercovitz[3]
[1]3D EAU, 21 rue Jacobi-Netter 67200 Strasbourg, France
[2]ENGEES-ICube, 1 quai Koch 67000 Strasbourg, France
[3]EDF R&D, LNHE - Groupe SIMPHY(P74), 6 Quai Watier 78401 CHATOU Cedex, France
E-mail: matthieu.dufresne@3deau.fr

Abstract: *This study presents a comparison between physical modelling and computational fluid dynamics (CFD) for investigating ecological continuity of the Poutès dam modification project. Water depth and velocity measurements have been carried out in the whole physical model. A CFD model has been built based on the geometry of the physical model. Simulations were performed using the OpenFOAM software and the InterFoam solver. Water depths and velocities have been extracted from the numerical model and compared to measurements. . The agreement is very good for water depths and quite good for velocities.*

Keywords: *Ecological continuity, computational fluid dynamics, physical modeling.*

1. Introduction

When a new dam is built or an old one is modified, ecological continuity is generally based on the construction of fish bypasses such as fish-ways or fish elevators (upstream migration) and downstream migration systems. These systems must be passable (fishes that are at the downstream side of a fish-way must be able to access to the upstream side) but also attractive and accessible (fishes have to find the fish-way and enter the fish-way before using it). This means that attractive velocities and water levels must be encountered in the downstream environment of a fish-way. This is also needed upstream to ensure that fishes will find the downstream migration system. To check this, physical modeling and numerical modeling are two possible approaches.

Physical modelling has been used since the beginning of the twentieth century to observe and measure the flow at small scale in order to investigate the behavior of a full-scale system before its building (Langhaar 1951, Ettema 2000). It is therefore a very useful tool to test and optimize a hydraulic structure. Nevertheless, investigations in a physical model take time, especially when a large number of configurations has to be tested in an optimization process. Moreover, there can be some difficulties associated with scaling effects: the choice of the Froude similarity leads to underestimate the turbulence influence in the physical model; surface tension effects can become significant for small-size models; results obtained for some specific phenomena such as air entrainment must be taken very carefully.

Computational Fluid Dynamics (CFD) has been developed for more than half a century (see Versteeg and Malalasekera 2007); pioneering simulations involving CFD were probably performed in the T3 Group at Los Alamos National Laboratory at the end of the 50's. Today, CFD is often used for hydraulic structures such as water intakes (Khan et al 2004), spillways (Chanel and Doering 2008), reservoirs (Rengers et al 2016)… A comparable study of the present work (but at a lower scale) has been carried out by Bayon et al. (2016). The main advantage of this tool is the rapidity for testing different configurations in an optimization process. Nevertheless, a number of physical phenomena such as sediment transport (bed-load transport, erosion, etc.) are not well known and still require a physical approach.

This paper presents a comparison between physical modelling and CFD for the modification project of the Poutès Dam built across the Allier River, France. The objectives of the study are to validate CFD against experimental data collected in the physical model of the dam (water levels, velocity fields) and to highlight the complementarities between the two approaches.
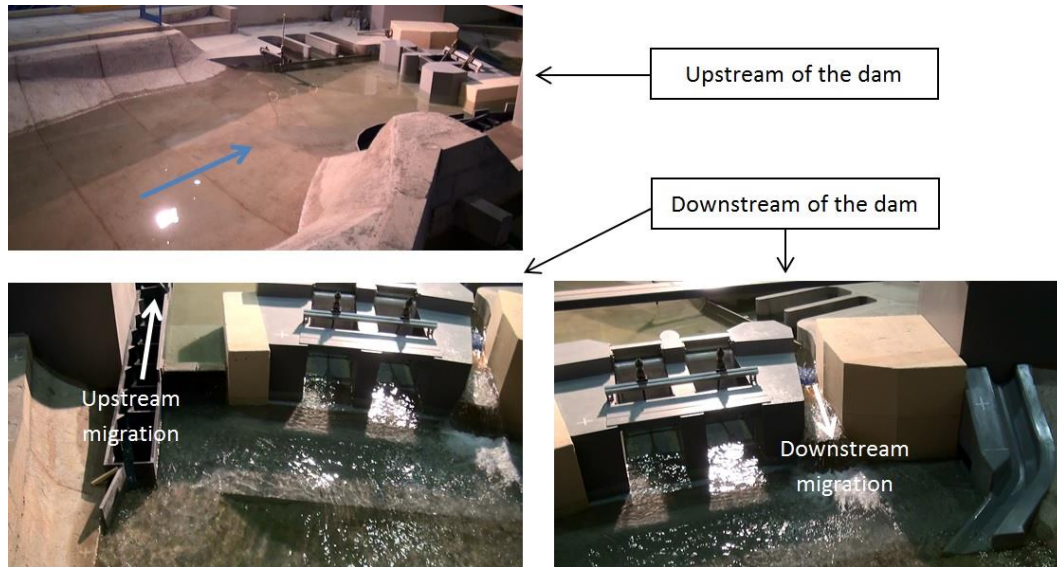
Figure 1. Views of the physical model

2. Material and Methods

2.1. Physical Model

A physical model of the Poutès reservoir project has been built by EDF; the physical model has been designed based on a Froude similarity, as generally done for models involving free surface flows. It is illustrated in Figure 1. The physical model includes the upstream reservoir, the dam itself, the fish migrations systems and the downstream region of the dam (in order to investigate the attractiveness of the water stream). The scale of the physical model is 1:13. It should be noticed that the geometry used for the physical model is different from the geometry finally chosen for the modification of the full-scale works (modification of the project in 2017).

Two sets of experimental measurements have been carried out. Firstly, seventeen locations have been chosen to measure the water level in the physical model using ultrasonic sensors (see Figure 2 for the locations of the measurements). The uncertainty of these measurements is about 1 mm. Secondly, Large-Scale Particle Image Velocimetry (LSPIV) has been carried out in order to measure the surface velocity field upstream and downstream of the dam. These measurements have been carried out using the free software FUDAA-LSPIV (developed by EDF and IRSTEA). The uncertainty of these velocity measurements has been roughly evaluated to 10% (on the basis of comparisons carried out on previous studies). Illustrations of the velocity measurements are given in Figures 7 and 8.
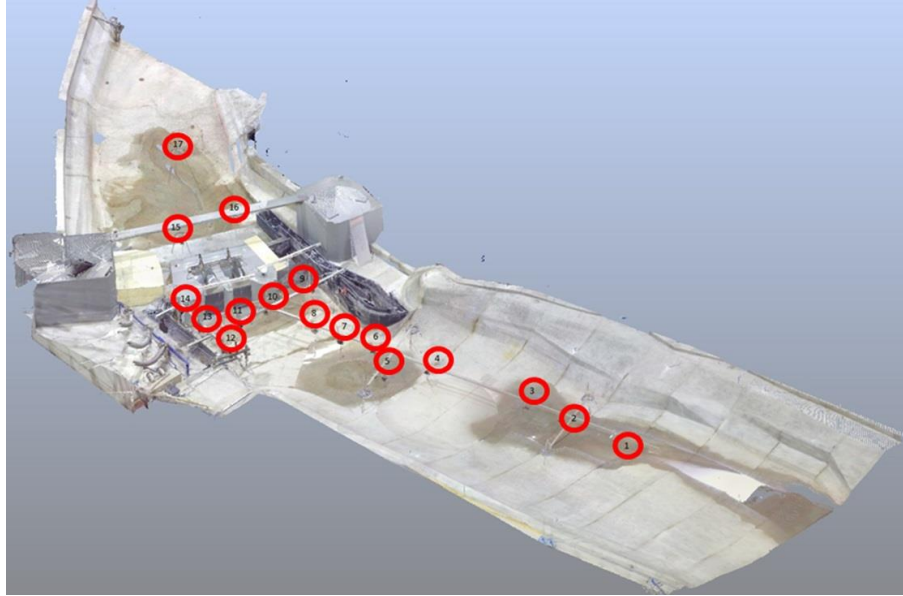
Figure 2. Locations of the water level measurements

The hydraulic conditions of the test chosen for the comparison between physical model and CFD model are given in Table 1. This experimental test has been chosen because it involves complex hydraulic behaviors such as instabilities in the velocity field and stationary waves on the free surface (see below). This choice is motivated by the objective of validation of a CFD approach for modelling the flow upstream and downstream of a dam in complex conditions.

The inlet discharge (116.5 l/s) and the discharge going to the water intakes (29.5 l/s) are boundary conditions whereas the other ones are results of the test. A weir regulates the water level downstream of the physical model in order to model the downstream influence of the river.

Table 1. Hydraulic conditions

	Physical model 1:13 (l/s)	Full-scale size (m³/s)
Downstream migration system	11.5	7.0
Left flap valve	32.8	20.0
Right flap valve	24.6	15.0
Upstream migration system (fish-way)	1.6	1.0
Attractive discharge (flap valve)	16.4	10.0
Water intakes	29.5	18.0
TOTAL	116.5	71.0

2.2. CFD Model

A CFD model has been built based on the geometric measurement of the physical model (three-dimensional scan, see Figure 2). Figure 3 highlights some views of the mesh (in this figure, the mesh is only shown on the walls of the domain). The mesh of the CFD model is composed of 3,400,000 cells. A grid sensitivity analysis has been carried out in order to check that this mesh was sufficient; the Grid Convergence Index (GCI) of the water depths is about 5% following the methodology of Roache (1994); the GCI corresponds to the numerical uncertainty for a 95% confidence interval (see Roache 1994).

The software OpenFOAM and the solver InterFoam have been used for the simulation of the flow. InterFoam is a solver dedicated to turbulent flows of two immiscible phases (water and air) based on the Volume of Fluid approach. The influence of the turbulence on the flow field has been modelled by the k-omega SST turbulence model. Even if this model is a simple and standard turbulence model (compared to Reynolds Stress Models or Large Eddy Simulation for example), we believe that it is sufficient to simulate the velocity field for this project because the flow is mainly influenced by the geometry rather than by the turbulence itself (we are not looking for Prandtl 2nd type secondary currents). Standard wall functions have been used for the rigid boundaries of the model; the roughness has not been calibrated and has been set to zero, which corresponds to a smooth boundary condition. This choice is justified by the small contribution of friction losses (relatively small velocities and small distances) compared to local losses (due to the perturbation of the streamlines). This point is very important because the calibration of the roughness can be an important difficulty for numerical models involving long distances.
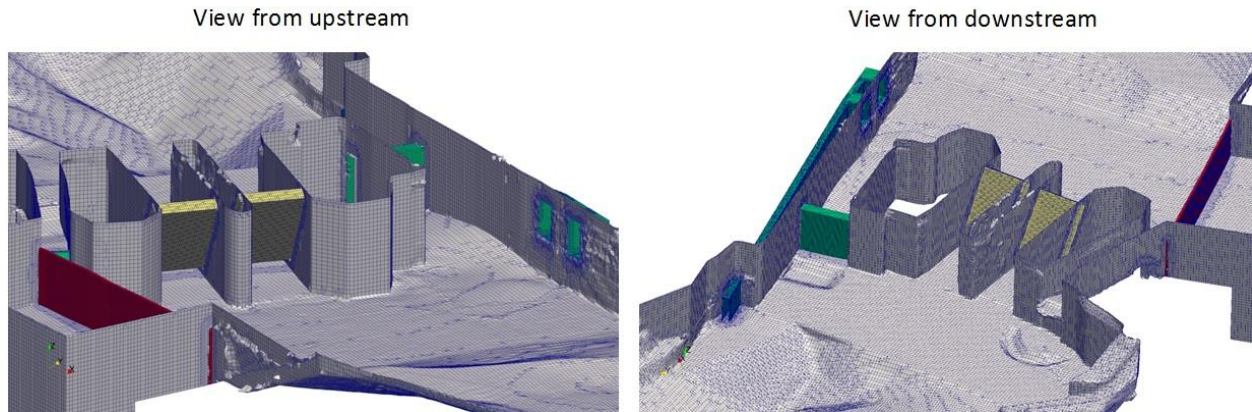

Figure 3. Views of the mesh used for the CFD model

The boundary conditions are illustrated in Figure 4: the inlet is modelled as a discharge coming from the bottom (as it is in the physical model) and the outlet is defined as a water level in order to model the weir located downstream of the physical model. The top of the model is defined as an atmospheric pressure.
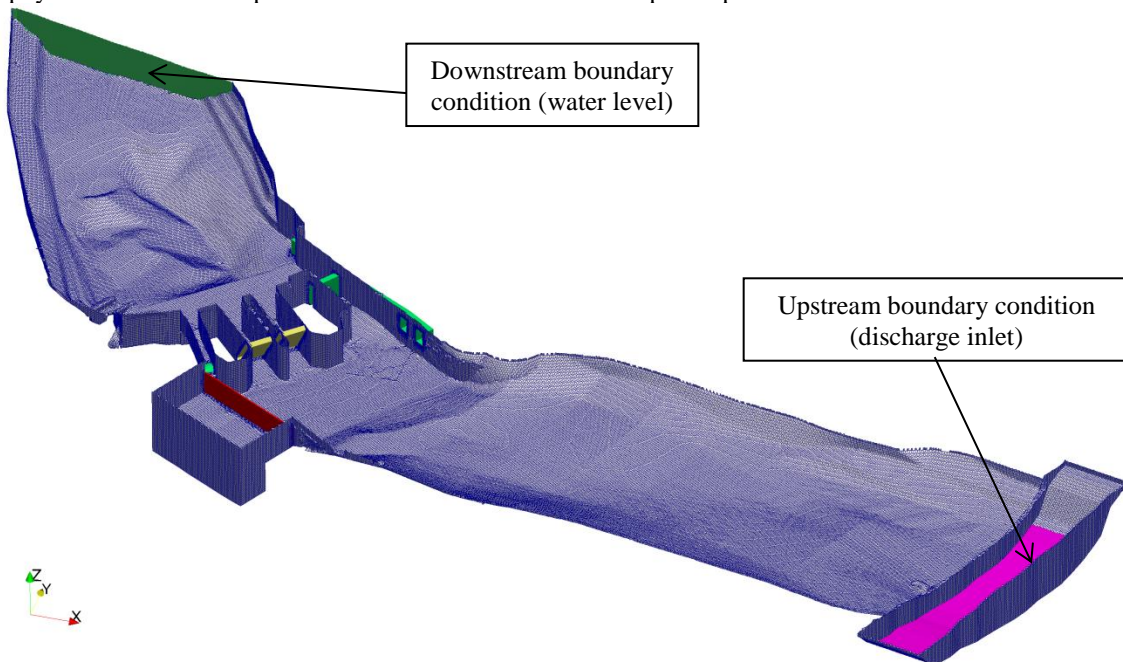

Figure 4. View of the boundary conditions used for the CFD model

The grid separating the reservoir and the water intakes (in red in the left part of Figure 3) has been modelled by a porous medium in order to avoid an excessive number of cells in this zone that is not very important for the main flow; the porosity has been chosen based on the size of the grid and the local losses formula adapted to the geometry of the grid.

Water depths and velocity fields have been extracted and compared to those measured in the physical model for the same hydraulic conditions (given in Table 1).

3. Results

3.1. Water Depths

Figure 5 shows a view of the numerical free surface in the whole model. The value z = 0 corresponds to the downstream water level. This figure highlights the stationary waves (already discussed before) upstream of the dam. This was one of the reasons why this test was chosen for the comparison between experimental and numerical results. Indeed, this kind of behavior is a challenge for the CFD model.
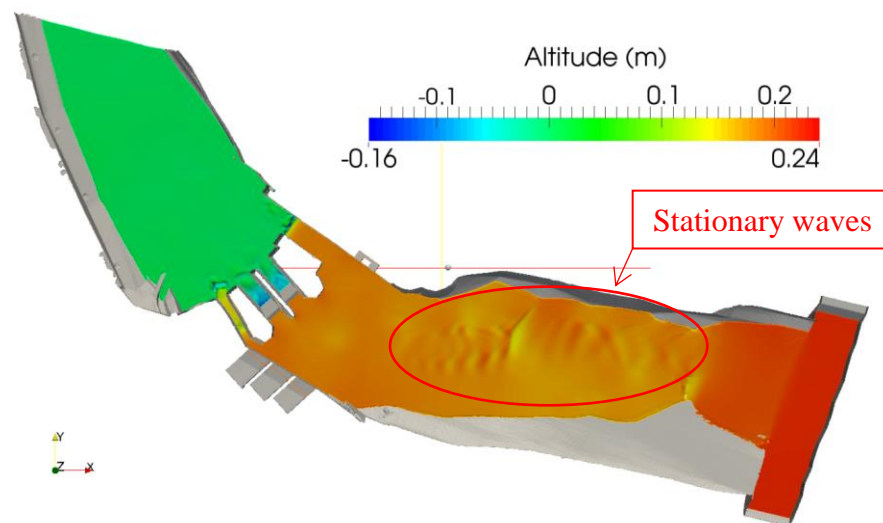

Figure 5. View of the free surface in the whole model.

As observed and measured in the physical model, the numerical results highlight important fluctuations of the water level. Figure 6 illustrates the relative fluctuations of the water depth $\Delta h/h$ at each measurement point (where Δh is equal to half the difference between the maximum water depth and the minimum water depth). The mean fluctuation is equal to +/- 10 mm (4%). Measurement point n°4 highlights a very large fluctuation of +/- 29 mm (21%), which can be explained by the position of this measurement point in the center of the zone highlighted in red in Figure 5.

Figure 7 shows the comparison between the measured and the simulated water depths; this figure shows that the agreement is very good. The mean discrepancy is 2% with a maximum up to 38% for measurement point n°4 (see the previous remark for this point).
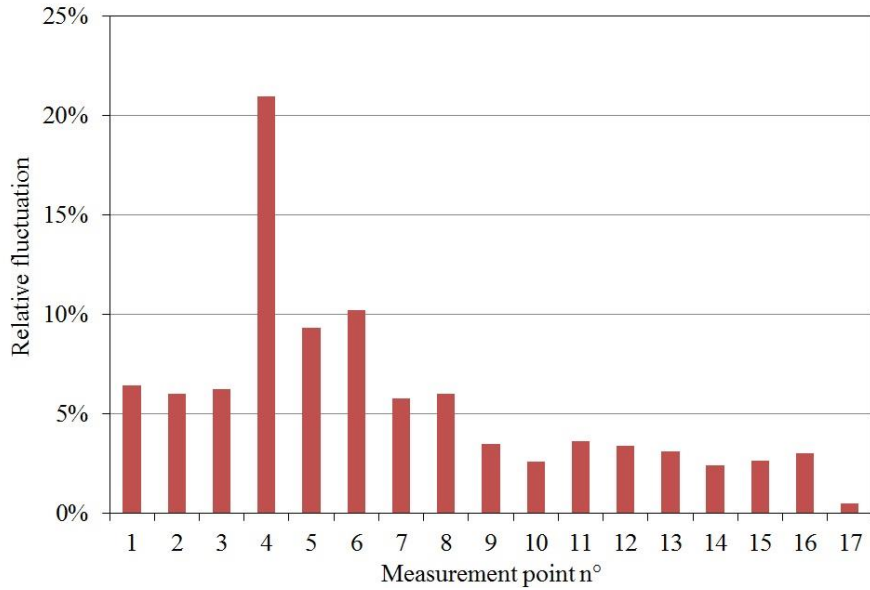
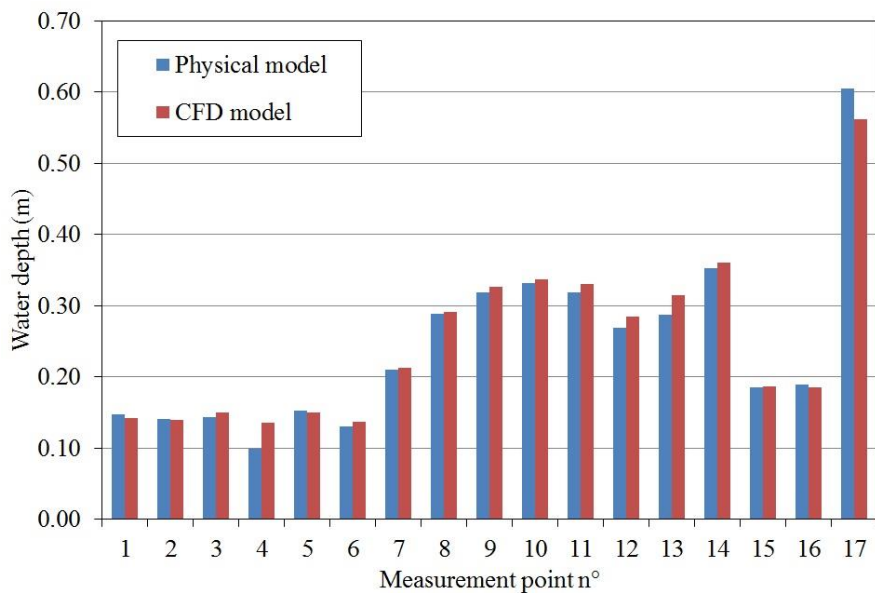**Figure 6.** Relative fluctuations of the water depths in the CFD model

Figure 7. Comparison between measured and simulated water depths

As a conclusion, it can be said that the free surface is well reproduced by the CFD model with an accuracy of about 5% (with the exception of points located in the zone perturbed by stationary waves).

3.2. Velocities

As for the water levels, the CFD model highlights some instability in the flow field. Indeed, the streams are not completely stationary but oscillate with time.

Figure 8 and Figure 9 illustrates the comparison between the surface velocity field measured by LSPIV and the surface velocity field calculated with the CFD model. The flow field is well described by the numerical model (same behavior, same recirculation zones, etc.) but the difference between numerical simulations and measurements is a little bit more important than for the water depths (approximately 20% to 30%). For example, the mean surface velocity of the upstream main jet is about 0.50 m/s in the physical model (0.68 m/s for the maximum velocity)

whereas it is about 0.60 m/s in the numerical model (0.90 m/s for the maximum velocity). In the downstream region of the dam, the mean velocity of the main jet is about 0.40 m/s for both physical and numerical models; the maximum velocity is about 0.52 m/s in the physical model whereas it is up to 0.70 m/s in the numerical model.

This higher discrepancy of the surface velocities compared to the water depths can be partly explained by the instabilities of the flow (oscillations of the streams). Nevertheless, we believe that the difference is not so important compared to the uncertainty of the velocity measurements (roughly evaluated to 10%).
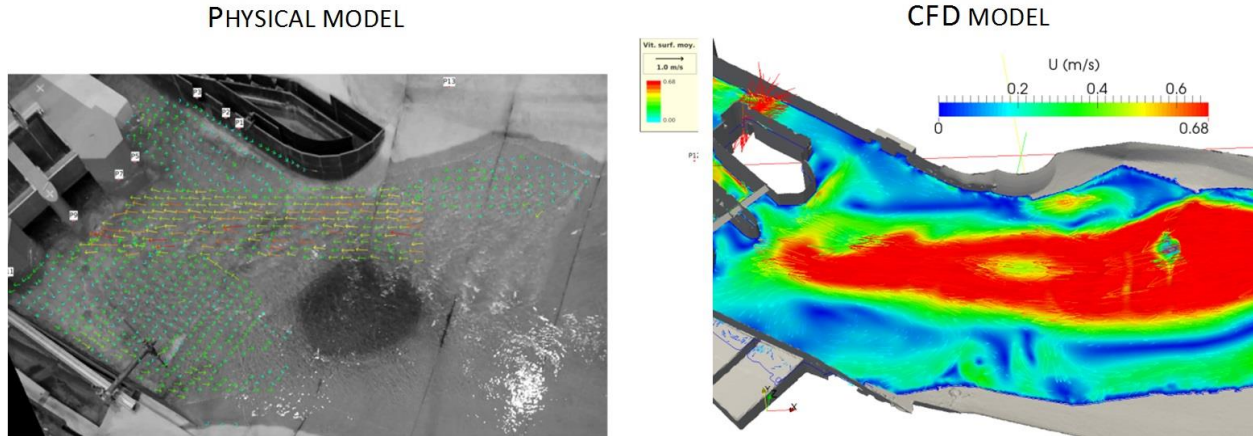

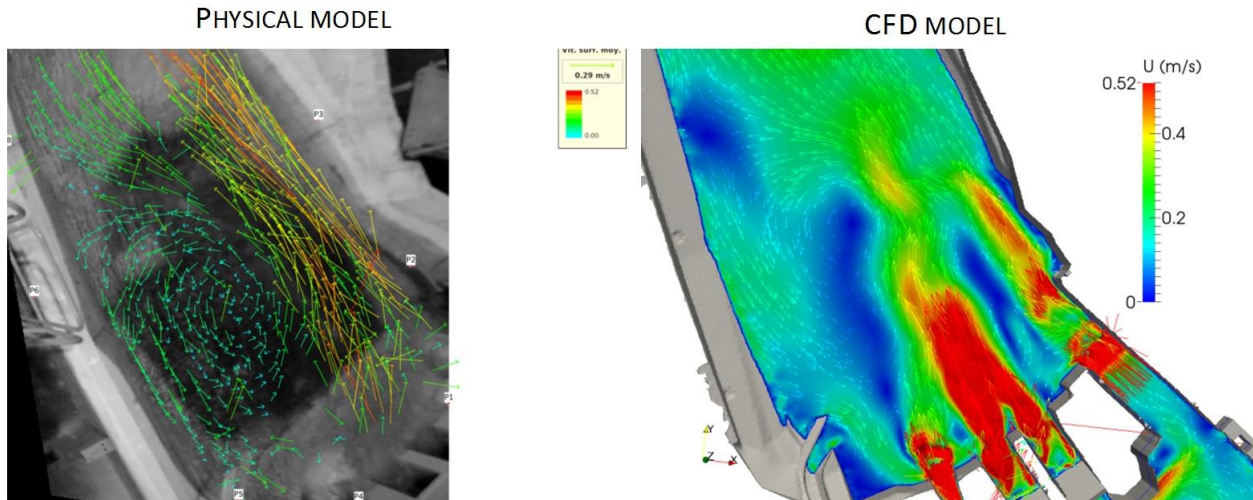**Figure 8.** Comparison between measured and simulated velocities (upstream)

Figure 9. Comparison between measured and simulated velocities (downstream)

4. Conclusions

The following conclusions can be drawn at the end of this study:

- The CFD model (without any calibration) shows very good agreement with the experiments for the water depths (about 5%) and relatively good agreement for the velocities (up to 20% to 30%).

- This kind of CFD models can be used to easily and rapidly test a number of configurations before testing the best one in the physical model; there is, therefore, a good complementarity between the physical modeling and the numerical modeling. It should be emphasized that the CFD model does not require any calibration.

- This kind of CFD models can be used to collect information in zones where measurements are difficult.

- Finally, this kind of CFD models can be used at scale 1 (without any scale effects).

- A coupled approach between physical and CFD models can therefore be used for the upcoming projects to ensure ecological continuity.

5. Acknowledgements

6. References

Bayon, A., Valero, D., García-Bartual, R., Vallés-Morán, F.J., & López-Jiménez, P. A. (2016). Performance assessment of OpenFOAM and FLOW-3D in the numerical modeling of a low Reynolds number hydraulic jump. *Environmental Modelling & Software*, 80, 322-335.

Bombardelli, F.A., Meireles, I., & Matos, J. (2011). Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways. *Environmental Fluid Mechanics*, 11(3), 263-288.

Chanel, P.G., and Doering, J.C. (2008). "Assessment of spillway modeling using computational fluid dynamics." *Canadian Journal of Civil Engineering*, 35(12), 1481-1485.

Dewals, B.J., Kantoush, S. A., Erpicum, S., Pirotton, M., & Schleiss, A.J. (2008). Experimental and numerical analysis of flow instabilities in rectangular shallow basins. *Environmental Fluid Mechanics*, 8(1), 31-54.

Dufresne, M., Dewals, B.J., Erpicum, S., Archambeau, P., & Pirotton, M. (2011). Numerical investigation of flow patterns in rectangular shallow reservoirs. *Engineering Applications of Computational Fluid Mechanics*, 5(2), 247-258.

Ettema, R. (2000). Hydraulic Modeling: Concepts and Practice, ASCE.

Khan, L.A., Wicklein, E.A., Rashid, M., Ebner, L.L., and Richards, N.A. (2004). "Computational fluid dynamics modeling of turbine intake hydraulics at a hydropower plant", *Journal of Hydraulic Research*, 42(1), 61-69.

Langhaar, H.L. (1951). Dimensional analysis and theory of models, John Wiley & Sons, New-York.

Savage, B.M., & Johnson, M.C. (2001). Flow over ogee spillway: Physical and numerical model case study. *Journal of Hydraulic Engineering*, 127(8), 640-649.

Rengers, E.E., da Silva, J.B., Paulo, P.L., Janzen, J.G. (2016). "Hydraulic performance of a modified constructed wetland system through a CFD-based approach", *Journal of Hydro-environment Research*, 12, 91-104.

Roache PJ (1994). Perspective: a method for uniform reporting of grid refinement studies. *Journal of Fluids Engineering* 116:405-413.

Valero, D., & Bung, D.B. (2015). Hybrid investigations of air transport processes in moderately sloped stepped spillway flows. In E-Proceedings of the 36th IAHR World Congress.

Valero, D., & García-Bartual, R. (2016). Calibration of an air entrainment model for CFD spillway applications. In Advances in Hydroinformatics (pp. 571-582). Springer Singapore.

Versteeg, H.K., and Malalasekera, W. (2007). An introduction to computational fluid dynamics, Pearson, Prentice Hall.