MODELLING FLOW UNDER BAFFLE SLUICE GATES

Pierre Le Faucheux[1], David Dorchies[2], Xuefang Li, Souha Gamri, Cyril Dejean, Séverine Tomas[3], Gilles Belaud[4]
[2]david.dorchies@irstea.fr,[3]severine.tomas@irstea.fr,[4]gilles.belaud@supagro.fr

Irstea UMR-GEAU, 361 Rue Jean François Breton, 34196 Montpellier

Phone: +33 (0)4 67 04 63 00, Fax: +33 (0) 4 67 16 64 40

KEY WORDS

Free surface turbulent flow, Numerical modelling, OpenFOAM, inclined sluice gate, VOF, RSM

ABSTRACT

The characteristics of the flow under a sluice gate are investigated experimentally. Velocities and pressure profiles as long as the contraction coefficient are measured for various flow rates (from 30m^3.h^{-1} to 50m^3.h^{-1}) and used to validate numerical simulations. In this paper, 2D and 3D simulations are conducted with the open source software OpenFOAM. Reynolds Stresses Model is used to model the turbulence and volume of fluids method to track the interface.

1. INTRODUCTION

Hydraulic structures are essential for water level and flow control in all waterways. Among these structures, vertical gates are very common, and for this reason they are the subject of many studies in experimental and Computational Fluid Dynamic (CFD), in order to derive stage-discharge relationships and study the effect of submergence on contraction coefficients (*Cassan and Belaud, (2012) [4]*). An engineering problem with vertical sluice gates is the force needed to change their opening. In this regard, radial gates are an interesting alternative to vertical gates, since the force needed to lift the gate is much lower. However, the contraction coefficient under such gates largely depends on the gate opening, whereas it is much more constant for vertical sluice gates. Such as for vertical gates, the effect of submergence still requires attention. Previous works showed that a key feature is the inclination of the lower lip of the gate, that is, the radial gate problem could be assimilated to the one of the inclined gate (*Gentilini, (1947) [5]*). *Belaud et al. (2014) [3]* used potential flow assumptions to derive the pressure force and the contraction coefficient under inclined gates, from which stage-discharge relationships can be derived in both free flow and submerged flow. But real fluid effects are not considered in potential flow needed to be quantified.

The improvement of CFD methods allows simulating complex situations with reasonable accuracy and computational time. It has been used in particular for vertical sluice gate (*Akoz et al., (2010) [1]; Cassan and Belaud, (2012) [4]*). Due to material limitation, potential theory are firstly used to describe the flow. As shown by *Kiczko & Al [6]*, it can be far enough if the situation is close to potential hypothesis but these conditions do not allow to model any kind of friction or interface. In a second step, methods using viscous fluids theory and an interface, enable to model a multiphase flow. It allows more challenging situations and predicting fluid's motions unreachable until then. Applied to vertical sluice gates, 2D computations enable the determination of contraction coefficient and pressure profiles.

Much less has been done for inclined gates, which is the core of this paper. Moreover, most of the numerical studies have been produced using well know commercial software such as ANSYS (Fluent), StarCCM or CFX. Since the beginning of 2000's, open source solution exists. Serious improvements have been done for the last 15 years and OpenFOAM now provides various solvers stable enough to compute reliable calculation. *Murcia [8]* and *Asim & Al [2]* have ensured the reliability of the solver with a comparison between Fluent and OpenFOAM. Also, most of the studies were performed with turbulence

---

1 Corresponding author

models k-ε or k-ω whereas *Cassan & Belaud [4]* shown that these models largely overestimated the contraction coefficient, while Reynolds stresses models should give more accurate results.

The main objective of this study is to investigate abilities to simulate water flow under an oblique sluice gate with OpenFOAM open source CFD software. The expecting outputs are flow velocities and pressure, as well as the contraction coefficient under inclined gate. Three dimensional effects need to be questioned too.

To validate the reliability of the simulations, numerical results are confronted with experimental data. After presenting the experimental setup and the simulation procedure, the results are presented and discussed, focussing on the comparison of flow fields, contraction coefficient and three dimensional effects.

2. MATERIAL AND METHOD

2.1. Experimental setup

A series of experiments has been performed in the hydraulic lab of Montpellier SupAgro. The objective is to explore the validity of the simulation model to parameterise energy momentum balance for inclined gates. The flume is 0.29 m wide rectangular channel, 8 m long and 0.40 m high. The sluice gate is tilted at 45 degrees and installed in the middle of the canal to allow the flow to be fully developed when hitting the gate. The opening of the gate is fixed at 50 mm for all flow rates. Four different parameters are measured: upstream velocity, pressure on the gate, upstream water level and the thickness of the contracted vein.

An Acoustic Doppler Velocity (ADV) is employed to get the velocity field with a sampling rate of 25Hz. The ADV measures the velocity in one point for 60 s, providing 1000 data by point. Measures are located at the centreline, 0.6 m ahead of the sluice gate, each centimetre from the ground. The height of the last point is limited by the ADV: it picks a velocity 30 mm below the instrument's head which has to be entirely submerged. To measure the pressure on the inside face of the gate, nine water column manometers are placed each centimetre on the centreline.

2.2. OpenFOAM

First, a two dimensional simulation is set. The size of the computational domain (around 8m long) suggest to start calculation under 2D assumption. However, the lateral walls effect and the development of a boundary layer might affect the flow even far from the wall. To investigate this, a three dimensional simulation is also set up. Due to the symmetry plan in the centre of the canal, only half of the whole domain is modelled. The initial conditions on the lateral wall are identical to the ones on the ground and boundary layers are built to insure a dimensionless wall distance y^+ defined by $y^+ = \frac{yu^*}{\nu}$ (where u* is the shear velocity, ν is the kinetic viscosity and y the wall distance) of the same order of magnitude (around 2.7).

OpenFOAM is an open source software used to simulate flow motion. It regroups various solvers depending on assumptions. Here, the solver interFoam is used. It is based on Reynolds Average Navier-Stokes (RANS) equations for two incompressible, isothermal and non-miscible fluids. This model presents a general formulation of the fluid mechanics:

$$\begin{cases} \frac{\partial u_i}{\partial x_i} = 0 \\ \rho u_j \frac{\partial u_i}{\partial x_j} = \rho K_i - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial^2 x_j} + \frac{\partial}{\partial x_i}\left(-\rho \overline{u_i' u_j'}\right) \end{cases} \qquad \text{(Equation 1)}$$

where *u* is the velocity, *p* the pressure, *K* the gravitational forces, μ the dynamic viscosity and $\rho \overline{u_i' u_j'}$ the Reynolds stresses.

Because interFoam is a transient solver and to reduce the effect of the fluctuation, the data will be averaged on the last 5 seconds of simulation in order to capture mean flow characteristic and to compare them to the corresponding experimental data which are averaged on the whole recorded sequence (30seconds).

2.3. VOF Method

In order to catch correctly the interface between the two fluids, interFoam uses Volume Of Fluids method (VOF). It defines a "fraction volume" α to separate the couple of phases. In the present case, the domain is filled with water and air which gives:

$$\alpha = \begin{cases} 0 & for \quad air \\ 1 & for \quad water \end{cases}$$

This parameter follows the transport equation:

$$\frac{\partial \alpha}{\partial t} + div(\alpha \vec{u}) = 0 \qquad \text{(Equation 2)}$$

Several models of turbulence are available in interFoam such as standard k-ε or k-ω (SST or not). A Reynolds Stress Model (RSM) developed by Speziale-Sarkar-Gatski (SSG) based on *Launder (1975) [7]* work is chosen in this case. RSM model solves transport equations for the Reynolds stresses and an equation for the dissipation rate to close the RANS equations. It leads to seven extra equations:

$$TimeDerivate + C_{ij} = D_{I,ij} + D_{L,ij} + P_{ij} + G_{ij} + \phi_{ij} - \epsilon_{ij} + F_{ij} \qquad \text{(Equation 3)}$$

where C_{ij} is the Convection term, $D_{T,ij}$ is the turbulent diffusion, $D_{L,ij}$ is the Molecular Diffusion, P_{ij} is the stress production, G_{ij} the buoyancy Production, Φ_{ij} the pressure strain, ε_{ij} the dissipation ad F_{ij} the production created by system rotation.

This model is supposed to better account for swirl, rotation effects or the rapid changes of strain than the one or two equations models. However, two important terms (Pressure-strain and dissipation rate) are partially unknown and can be difficult to represent correctly during the simulation.

2.4. Mesh, boundaries and initial conditions

The upper side of the computational domain is located 1.4 m above the ground to insure that no side effects disrupt the water flow. Otherwise, to minimize the difference due to scale effects, the model and the original canal have the same dimensions. A gate is placed 4 m downstream to the entrance, oblique at 45 degrees, letting an opening of 50 mm.

The origin of the coordinate system is placed on the left side, on the ground at the vertical of the opening; direction-x being the direction of the flow, “y” the height of the canal and “z” the depth.

To accelerate convergence, a power-law profile with exponent 1/6 is introduced at the inlet for the water part. Velocity of the air is set to be 10^{-7} $m.s^{-1}$ in x-direction. No-slip conditions are applied to the ground and the sluice gate (and to lateral wall in 3D) which results in $u = v = w = 0$. Rest of the boundary conditions are summed up with the general view of the domain in **Figure** 1.

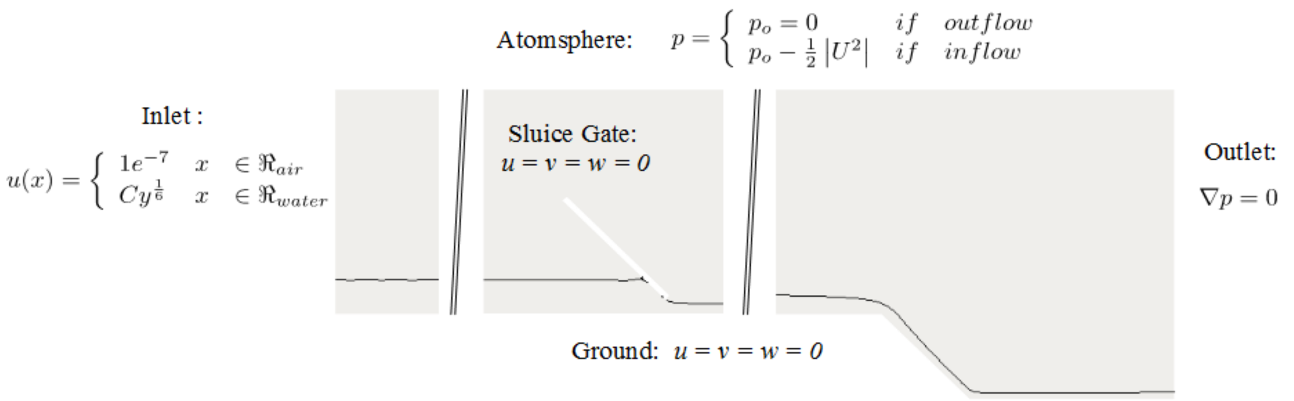

Figure 1: General view of the model and Initial conditions

Case	Q30	Q40	Q45	Q50
Q ($m^3.h^{-1}$) (experimental)	30	40	45	50
U($m.s^{-1}$) (experimental)	0.413	0.428	0.395	0.386
k ($m^2.s^{-2}$) (calculated)	0.00753	0.0010	0.0117	0.0147
Length of the turbulence (m)	0.0025	0.0025	0.0025	0.0025
Intensity of the turbulence (%) (experimental)	5.42	6.14	7.05	8.10
ε ($m^2.s^{-3}$) (calculated)	0.00074	0.0012	0.00143	0.00202
Upstream Water level (cm) (experimental)	6.95	8.95	10.9	12.4

Table 1 Boundary condition

To force the flow to setup quicker, the volume is initialized at the start. It means the upstream water level is set to the upstream level measured from the experiment, and downstream, water level follows an adaptive curve. As it is done for the inlet, the entire volume of water is set with velocity power law profile in 1/6. The intensity of the turbulence is taken from the experiment (see **Table** 1) and the length of the turbulence is decided to be 5% of the opening gate. Initial value of turbulence kinetic energy and turbulent dissipation are calculated from the previous parameters. **Table** 1 regroups the initial conditions used here.

To reduce computational time, the sluice gate is wrapped into a hexagonal mesh. In order to enhance results and due to the turbulence model chosen, a particular attention is given near walls. Boundaries layers are placed around the ground, the sluice gate and the lateral wall. This prevents wall functions to be used and allows the equations to be solved even in the near wall regions (**Figure** 2).

The pertinence of the simulations is highly influenced by the mesh density (*Akoz & Al. (2010) [1]; Oner & Al. (2012) [9])*. To ensure that results are independent of the grid, a mesh convergence test is performed. Three different meshes have been tested. They only differ by the number of cells in the zone filled with water. The part filled with air is unchanged for the three meshes. The coarser mesh contained 300 000 cells, the intermediary 400 000 and the finer 470 000. The respective minimal volume of cells are $1.30x10^{-5}$ m^3, $1.08x10^{-5}$ m^3 and $8.54x10^{-6}$ m^3.

Time required to converge greatly depends on the machine used. With 8 cpu machine, the difference is significant. Around 50 additional minutes are required for the finer mesh to compute 1s. However, this difference drops to 10 minutes when calculation is done with 24 cpu. According to **Figure** 3, the intermediary mesh seems to provide enough refinement. The behaviour of the velocity profile computes with intermediary and finer is similar, (1% difference at most). However, the small additional amount of time required and larger stability observed leads to consider the finer mesh for calculation.

To allow to use RSM turbulence model without any wall functions, mesh has to be refined close to the wall. This refinement can be quantified by the dimensionless wall distance y^+. Two value of y^+ have been tested here: 1.77 and 2.69. Established velocities differ by only 1.14% between the couple of cases.

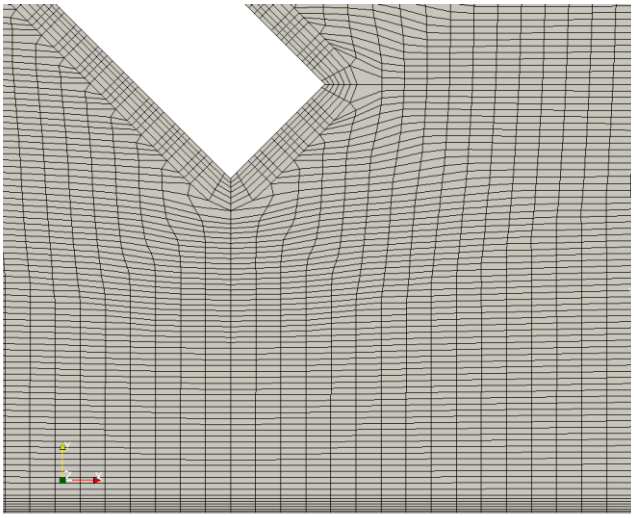

Figure 2: Boundary layers at right bottom of the gate for an opening of 50 mm

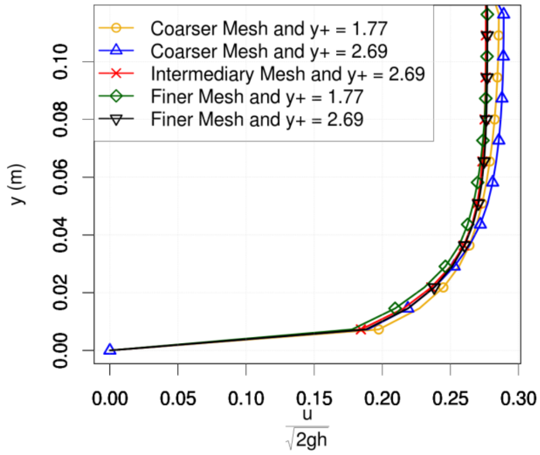

Figure 3: Mesh convergence test (vertical velocity profile for water flow rate of 50 $m^3.h^{-1}$)

3. RESULTS

The problem is first boarded with a two dimensional approach modelling the centre plane of the canal. Velocity profiles, pressure along the gate and contraction coefficient downstream the gate are compared to measures. Velocities profiles are normalized by $\sqrt{2gh}$ where “g” is the gravity and “h” the draft.

3.1. Velocity Fields and pressure

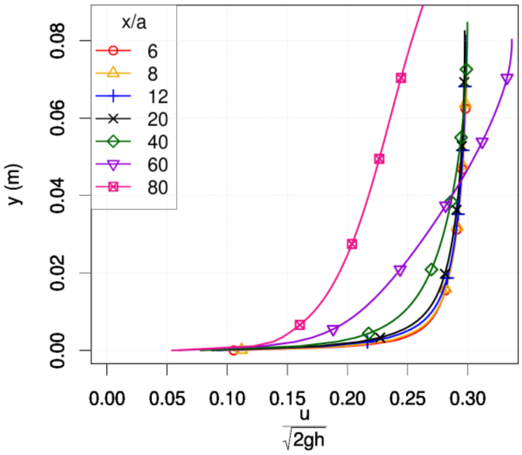

Figure 4: Simulated upstream velocity profiles at several abscissa (where a is the opening 50mm)

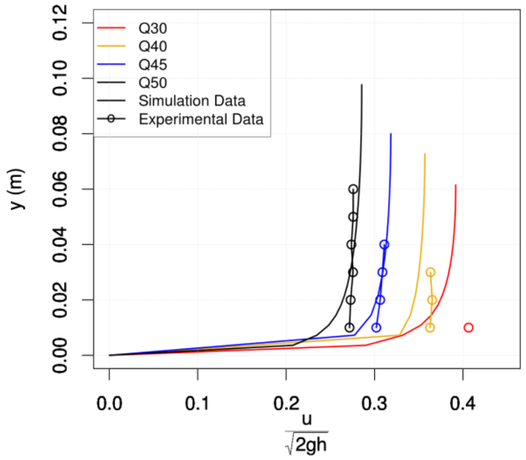

Figure 5: Comparison of vertical velocity profile numerical and experimental for four water flow rate

Figure 4 illustrates the upstream velocity at different location defined by the abscissa of each profile divided by the gate opening. Velocity seems established at 1 m before the gate and relative gap with the maximum velocity is minimal (0.5% at the most).

Figure 5 shows the mean velocities obtained upstream of the gate for each scenario. Simulated velocity profiles deviate from the experimental results in the near wall region. It seems that the turbulence model introduces too much energy losses and slows the establishment of the velocity profile. This difference increases with the reduction of the inlet flow rate. Even in the zone far enough from the wall, the velocity profile does not match the experiments. Gap varies from 0,85% for the highest flow rate to 14,5% for the smallest. Nevertheless, some precautions should be taken when the comparison is made with the smallest flow rates, as the number of experimental data available is limited (only one measurement for the smallest one).

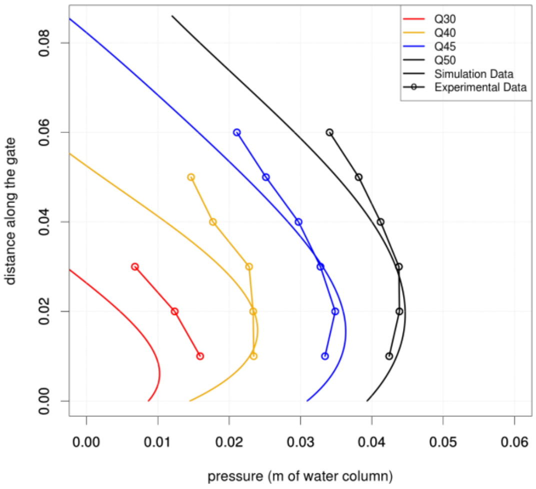

Figure 6: Comparison of pressure profile, along the gate, numerical and experimental for four water flow rate

On **Figure** 6, the distance from the bottom of the gate is plotted against the total pressure. Due to calculation method, total pressure given by OpenFOAM differs from the experimental data. Calculating the sum of the static and dynamic pressure, give results much closer to the observations. Indeed, the relative difference, between the maximum experimental pressure and the one obtain directly from interFoam is stable around 45%. In the case where the pressure is post-treated from the static and the dynamic pressure, this difference drop to 28% for 30 $m^3.h^{-1}$ and below 6% for the rest of the flow rate.

3.2. Contraction Coefficient

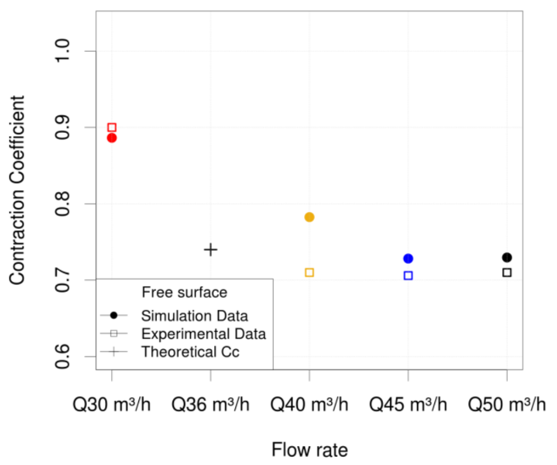

Figure 7: Comparison of numerical and experimental contraction coefficients for four water flow rate

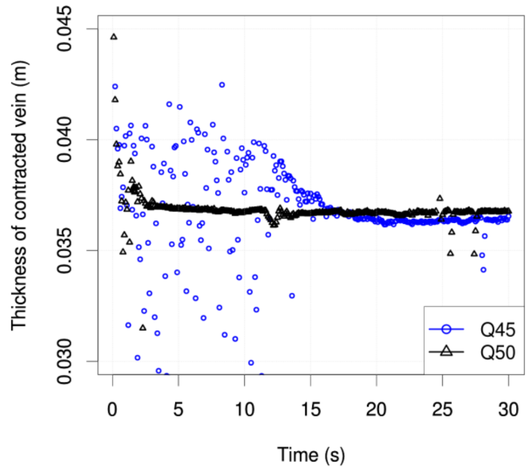

Figure 8: Evolution of the vein's thickness along the time for Q = 45 $m^3.h^{-1}$ and Q = 50 $m^3.h^{-1}$

Figure 7 shows the results for the contraction coefficients. The contracted section is obtained at 0.20 m, which corresponds to about 4 times the gate opening. This is consistent with experimental measures. The range of relative error between simulations and experiments is between 1.5% and 9.5%. Contraction coefficient seems constant for water flow rate higher than 45 $m^3.h^{-1}$ which is consistent with theoretical results obtained from potential flow assumption (*Belaud et al., 2014 [3]).* Simulation result for 40 $m^3.h^{-1}$ is surprisingly high compared to experimental and potential flow results. Such a deviation could be explained by interface detection issue.

3.3. Three dimensional effects

Data have been measured only on the centre line. If the wall effect on the ground is taken into account in the 2D simulation, the model does not account for the friction on the lateral wall, or for the vortices generated at the upstream corner from the gate. In order to investigate these effects, 3D simulations are performed with RSM. As mesh becomes heavier (around 5 million cells) and computational resources has been increased (96 cpu) allowing to simulate 1 second in 4 to 5 hours. The volume is initialized with the converged 2D solution. **Figure** 9 to **Figure** 12 regroup experimental and the numerical results from the two and three dimensional simulations for a flow rate of $50m^3.h^{-1}$.

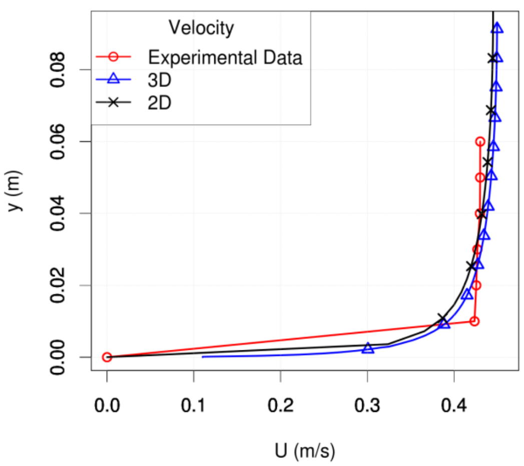

Figure 9: Comparison of velocity profile obtained from 2D, 3D and experimental data

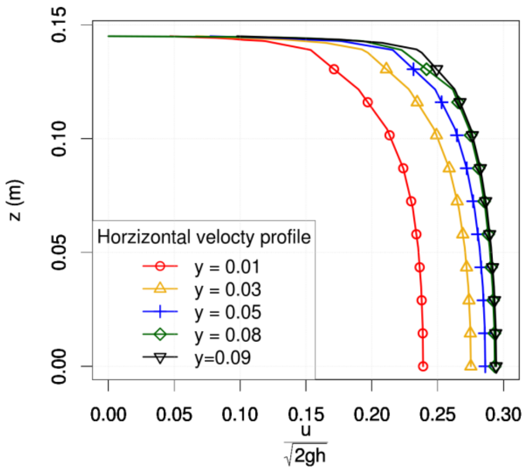

Figure 10: Horizontal velocity profile through the canal at 5 different heights

In the near wall region, velocity profile is getting closer to experimental data than the previous simulation. The mean velocities in the upper half remain higher than the measured ones in the 3D simulations. The horizontal profiles across the channel are depicted in **Figure** 10. Wall's effects are clearly visible up to a distance of about 0.1 m, leading to slightly higher velocity at the middle of the canal in 3D compared to the mean velocity in 2D. Velocity can be considered constant at z <0.05 m.

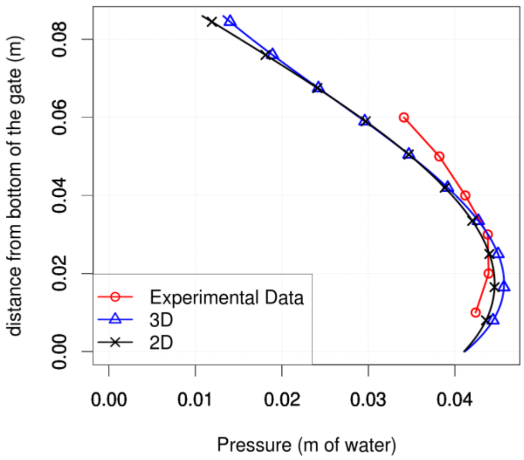

Figure 11: Comparison of Pressure profile derived from 2D, 3D and experimental data

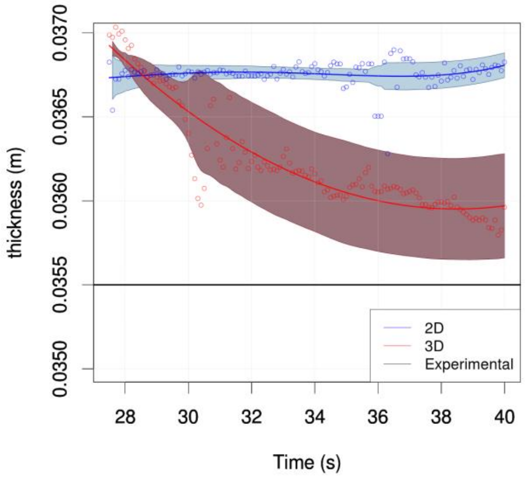

Figure 12: Comparison of contracted vein thickness for 2D, 3D and experimental data

No real distinction might be done between 2D and 3D simulations except for small increase of the maximum pressure. Therefore, the 3D simulation does not bring significant improvement on the pressure along the gate.

Figure 12 shows the contraction coefficient against time simulation in both case (2D and 3D) along the experimental one. A significant gap is observed at the end of the simulation with a decrease of error between simulation and measurement.

4. CONCLUSION

The objective of this study was to explore the real flow characteristics of water under an inclined sluice gate. Results show that CFD is a good alternative to experimental studies, although results are sensitive to modelling assumptions.

In the situation analysed here, the advantages of the 3D simulation over 2D, is not clear since the difference is very tiny for velocity profile and pressure along the gate. Moreover time required to perform 3D case is three to four times larger and more demanding on computational resources. In term of contracted coefficient, the 3D setup might be justified as the 2D overestimated the water level.

In order to explain the difference with experimental data, different assumptions should be explored such as the impact of the turbulence's intensity, of turbulent models or wall functions in the boundary layers.

However, 3D simulation may have a clearer advantage in complex situations where the flow is very swirly and contain many vortices, which happens on gates in some cases.

Finally, this study has shown that OpenFOAM is a good alternative to the commercial software, and it offers possibilities to compute easily a series of configurations that may facilitate the generation of simplified head-discharge-opening curves.

5. REFERENCES AND CITATIONS

Akoz M.S., Kirkgoz M.S., Oner A.A. (2010): *Experimental and numerical modelling of a sluice gate flow*. Journal of Hydraulic Research [1]

Asim T., Mishra R., Kaysthagir S., Aboufares G. (2015): *Performance Comparison of a Vertical Axis wind turbine using Commercial and Open Source Computational Fluid Dynamics based*, University of Huddersfield, [2]
Belaud G, Tomas S & Cheviron B 2014: *Contration under inclinate and radial gates*. 3rd IAHR Europe Congress, Book of Proceedings. [3]

Cassan L. and Belaud G. (2012): *Experimental and Numerical Investigation of Flow under Sluice Gates*, American Society of Civil Engineers [4]

Gentilini B, (1947). *Ecoulements sous les vannes de fond inclinées ou à secteur*. La Houille Blanche [5]

Kiczko B, Kubrak J, Kubrak E, *Experimental and numerical investigation of non-submerged flow under a sluice gate*, Annals of Warsaw University of Life Sciences – SGGW [6]

Launder, B. E., Reece, G. J. and Rodi, W. (1975*), Progress in the Development of a Reynolds-Stress Turbulent Closure.*, Journal of Fluid Mechanics, Vol. 68 [6], pp. 537-566. [7]

Murcia, O.E.H, *Comparison of OpenFOAM and Fluent for steady, viscous flow at Pool 8, Mississippi River*, Iowa Institute of Hydraulics Research, The University of Iowa. [8]

Oner A.A., Akoz M.S., Kirkgoz M.S., Gu-mus V. 2012: *Experimental Validation of Volume of Fluid Method for a Sluice Gate Flow*. Advances in Mechanical Engineering. [9]