Assessment of a numerical method to forecast vortices with a scaled model

G. Guyot[1], B. Huber[2] & A. Pittion-Rossillon[1]
[1] Electricité de France Hydro Engineering Center, Le Bourget du Lac, France Department or Institute,
[2]Vienna University of Technology, Institute of Hydraulic Structures, Vienna, Austria
E-mail: gregory.guyot@edf.fr and boris.huber@kw.tuwien.ac.at

ABSTRACT: Until now the fine representation of the complex hydraulic phenomena such as Vortices have been related mainly to physical modeling for industrial purposes. EDF has been testing the 3D CFD code "Flow 3D©" for free-surface flow simulation for seven years. Flow 3D is a commercial CFD Code using volume of fluid (VOF) method with a rectangular mesh. In this study it is assumed that flow is liquid water with an interface simulated by the VOF method. The main difficulty of using VOF methods to capture the water levels is the relevance and the stability of their numerical results. In the field of vortices, it has been decided to test how reliable Flow 3D is in this context.

This study aims at assessing the simplified method which has been proposed by the EDF Hydro engineering center [18][20]. This test has been based on the large scale model operated by the Institute of Hydraulic Structures which belongs to the Vienna University of Technology. To enable reproduction of development of air-entrainment vortices, the scale has been chosen equal to 1:18 under the classic criteria of 1:20. This scale model based on Froude similarity has been designed to predict the flow behavior during filling or emptying the Latschau Reservoir which is a part of the Montafon/Voralberg hydro-power plant project. One of the operating conditions has been chosen to analyze if the simplified method is reliable.

Keywords: Free surface vortex, physical scale model, CFD, Water intakes, VOF method

1 INTRODUCTION

This study was prompted by practical problems encountered in many situations of hydraulic engineering such as industry, irrigation and water power generation. Water is often pumped from free-surface basins or tanks by intakes. For some cases free surface vortices can appear and can entrain air into the intake. This phenomenon can cause operational troubles or damages for turbine, pumps or hydraulic networks, such as loss of performance, noise and vibration,[5][8][12][19]. Nowadays the most efficient way to avoid vortices is to ensure that the minimum water head is greater than the critical submergence head and to check that no vortex appears using physical models at reduced geometric scale.

Several investigations have been made to assess the critical submergence [4][7] and standards have been published [14]. For non conventional designs, these criteria can be considered inadequate and the use of a physical model is then necessary.

However it is unlikely that for some projects a physical model study with several intake geometries should be undertaken due to time and financial constraints.

Until now the fine representation of the complex hydraulic phenomena such as Vortex has been related to physical modeling for industrial purposes. But progress in the Computational Fluid Dynamics (CFD) field leads to use this tool for the vortex phenomenon [15][16].

EDF has been testing the three dimensional CFD code "Flow 3D©" for five years. This software is a commercial CFD Code, based on a Volume of fluid method with a rectangular mesh. The main difficulty of using finite volume programs is the relevance and the stability of their numerical results. EDF has decided to test in the field of vortex how reliable "Flow 3D©" is.

This study aims at assessing the simplified method which has been proposed by the EDF Hydro Engineering center [18][20]. The two first mentioned studies were carried out to build this method by testing sensitivities to the numerical parameters and to find the easiest and fastest way to forecast the risk of vortex.

In order to ensure that the method is a reliable way to detect the formation of vortices, a large scale model has been used. The model has been ordered by the Voralberger Illwerke AG to design a new pumped storage power plant in Voralberg, Austria. This scale model made by the Institute of Hydraulic Structures of the Vienna University of Technology deals with the operation conditions of an intermediate reservoir named Latschau reservoir.

The best operation case to assess the simplified method is a case of emptying the Latschau reservoir. For this case, the EDF- method has been fully applied from the large domain to the refined domain, to simulate the scale model. The following article shows the comparisons between the two approaches.

2 REMINDER OF THE SIMPLIFIED METHOD

Several vortex markers have been mentioned by different authors but only five have been selected by our study. The use of the free surface shape, the vorticity, and the Q criterion [12] are described in detail in this article. The Delta criterion [9]and the Swirling strength [13] have been also used in our investigations. To go straightly to our results the two last mentioned methods are not described in detail in the document.

A simplified method to predict the risk of free surface vortices at intakes with 3D CFD Code named Flow 3D has been presented in [20]. In order to check how reliable the numerical simulation is, the results of those calculations have been compared with a physical model. The experiment stand depicted in [18] and [20] has been designed to provide a full symmetric geometry of the reservoir and the inflow conduits.

The more accurate vortex marker is the Q criterion, which allows rapidly pinpointing and sharply sorting out the different swirling structures calculated.

2.1 *Q criterion*

Proposed by Hunt [12], this criterion can be evaluated specifically in an incompressible fluid. The vortex is described in a nutshell with the two following conditions:

In a vortex the rotational stress is stronger than the deformation one. It means that the second invariant of the velocity tensor is positive. It has to be highlighted that it is not a sufficient condition. If the second invariant of the velocity tensor was positive in a certain area it would not inevitably involve a vortex.

The pressure in the flow is smaller than the surrounding one. If the velocity tensor is written as follows, the rotation rate is:

$$\underline{\underline{\nabla}}\,\underline{v} = \begin{pmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{pmatrix} \tag{1}$$

u, v, and w are the three components of the velocity in Cartesian coordinates.

$$\underline{\underline{\Omega}} = \frac{1}{2}\left(\underline{\underline{\nabla}}\,\underline{v} - {}^{t}\underline{\underline{\nabla}}\,\underline{v}\right) \tag{2}$$

The strain rate would be written as

$$\underline{\underline{S}} = \frac{1}{2}\left(\underline{\underline{\nabla}}\,\underline{v} + {}^{t}\underline{\underline{\nabla}}\,\underline{v}\right) \tag{3}$$

The Q criterion would be

$$Q = \frac{1}{2}\left(\mathrm{tr}\left(\underline{\underline{\Omega}}.\ {}^{t}\underline{\underline{\Omega}}\right) - \mathrm{tr}\left(\underline{\underline{S}}.\ {}^{t}\underline{\underline{S}}\right)\right) \tag{4}$$

It is easy to analyze and speed up the calculation results with this criterion by sorting quickly the different types of flow area thanks to the Q value. Since the Q value is a scalar the axis system orientation is no matter.

Three steps are needed to clearly define the type of the vortex. Lastly, the value of the vortex predicted could be overestimated if the vortex intensity is high.

2.2 *Methodology and meshes*

The numerical method which has been selected is based on three steps. A sensitivity study has been performed on the meshes geometry to obtain the retained meshes.

2.2.1 *The possible vortex area pinpointed*

The first simulation to be performed is a large cell size simulation. In that step the entire study domain must be represented. This simulation aims at determining the area where a vortex can likely appear.

The results of this simulation enable to locate the area where Q is at its maximum. It is likely a vortex can appear there.

2.2.2 *Vortex appearance checking*

A second simulation with a refined mesh is needed to check the vortex existence. In that purpose a smallest area must be simulated. The mesh covers only the part of the domain where the Q value is a maximum.

The flow conditions imposed at the boundaries must be extracted from the previous simulation. If a vortex is confirmed it would be useful to know as accurately as possible the dimensions of this phenomenon.

2.2.3 *Evaluate the vortex shape*

To determine the shape of the vortex, a third simulation is needed. The calculation domain must be reduced to the close area located around the vortex.

The flow conditions imposed at the boundaries must be once again extracted from the previous simulation. The cell size must be strongly reduced to reach a very thin size.

2.2.4 *Main simulation parameters*

For both of the steps, the fluid is assumed as Newtonian. The calculations were done without any specific turbulence model (when the mesh gets finer, the simulation without a turbulence model tends towards a direct numerical simulation), but with a second order monotonicity preserving momentum advection. Those parameters have been chosen in accordance to the study [18], where a lot of sensitivity studies have been carried out. In a nutshell, to achieve the study [18], the k-e, the RNG and the LES turbulence model were tested but in this particular problem it appeared that the chosen turbulence model has minor influence.

3 LATSCHAU RESERVOIR SCALE MODEL

The context of large scale model has been described in detail in [21]. In the following chapter only the main aspect which concerns the vortex simulations are recalled.

3.1 *Description of the Latschau reservoir*

The reservoir is depicted in Figure 1. There are 4 structures in the basin: the first one is a conduit which connects the reservoir to another basin next to it. The second one, situated in the northern area of the reservoir, is the existing in-/outlet structure (called “2-in”). The third, new structure, “3-out” is projected in the southern area. Furthermore, there is a canal, from which water can also come into the reservoir.

Strictly speaking both structures “2-in” and “3-out” can either work as an inlet, leading water from the reservoir down to the power plant, or as an outlet, while pumping water into the reservoir. Concerning the development of air-entrainment vortices at the new structure, only the case when it works as an outlet is of interest here.

Generally speaking, vortices can develop when flow to an outlet is asymmetric. In our case, due to the shape of the reservoir, flow towards the new outlet is very asymmetric, varies with the water level and furthermore depends on the specific operating condition. The picture below shows an illustration of the topography of the basin (all heights mentioned refer to nature scale).

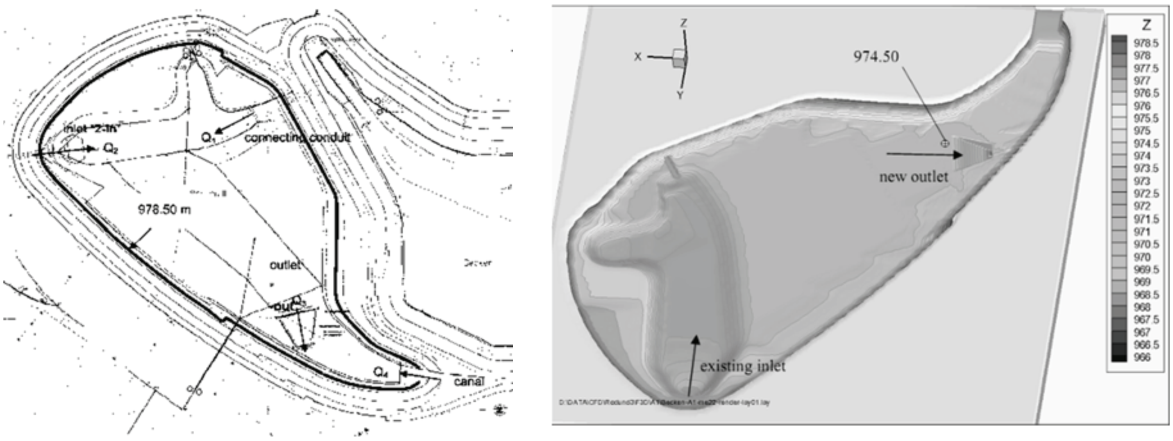


Figure 1 : Plan view and topographic illustration of the Latschau reservoir

3.2 *Physical model*

It is recommended by several researchers, e.g. Knauss [3], that a physical model should not be smaller than a scale of 1:20 in order to reproduce the development of air-entrainment vortices. The scale chosen here was 1:18. Another recommendation is to increase inflow velocity. To meet this concern additional model tests were conducted with increased discharges.

The physical model (based on Froude law of similarity had overall dimensions of 30 x 16 m. To build the model accurately, bricks were positioned along the major contour lines and leveled exactly. The two inlet/outlet structures made of acrylic glass, the conduit and the canal were placed at their positions. Then the model was filled with compacted sand and the surface concreted. The reservoir was built up to a level of 978.50 m which is 4 m above the surface of the reservoir.

The new outlet structure has a trapezoidal shaped ramp that leads from the reservoir surface at 974.50 m down to the bottom of the inlet at 967.00 m (see **Figure 2**). The drawdown-level of the reservoir is intended to be very low at 975.50, only 1 m above the reservoir surface. With such a small water cover between the water level and the outlet axis it is expected that air-entrainment vortices could develop.

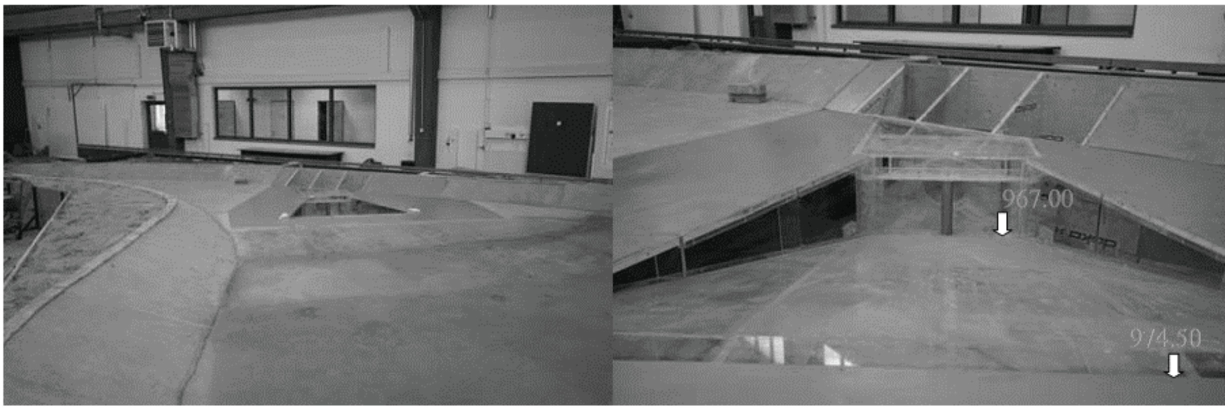


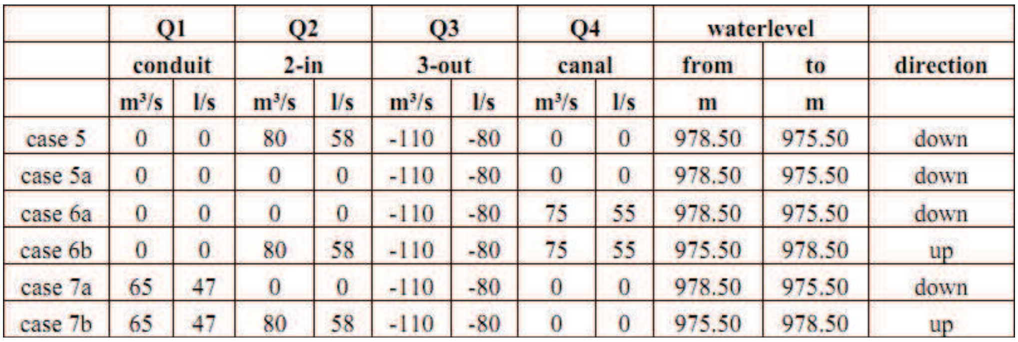
Figure 2 : View of the outlet "3-out"

3.3 *Operating conditions tested in scale model*

Disregarding cases when the new structure works as an inlet into the reservoir the following operating conditions were examined in the model tests (m³/s in nature, l/s in model scale; positive values mean flow into the reservoir) – Table 1.

The regular operating conditions are case 5 and 5a, the other cases are special operating conditions which occur only once a year. So the model tests concentrated on the regular cases. As mentioned before, additional model tests were conducted with increased discharges (up to 130%) in order to enforce the development of vortices.

Table 1 : Operating conditions tested on the scale model



	Q1		Q2		Q3		Q4		waterlevel		
	conduit		2-in		3-out		canal		from	to	direction
	m³/s	l/s	m³/s	l/s	m³/s	l/s	m³/s	l/s	m	m	
case 5	0	0	80	58	-110	-80	0	0	978.50	975.50	down
case 5a	0	0	0	0	-110	-80	0	0	978.50	975.50	down
case 6a	0	0	0	0	-110	-80	75	55	978.50	975.50	down
case 6b	0	0	80	58	-110	-80	75	55	975.50	978.50	up
case 7a	65	47	0	0	-110	-80	0	0	978.50	975.50	down
case 7b	65	47	80	58	-110	-80	0	0	975.50	978.50	up

3.4 *Results of the model tests*

Before a test was started the reservoir was filled up to the starting water level. The discharge at the inlets and the outlet was increased within approximately 20 seconds, starting from a quiet water surface, because that also enforced the development of vortices. The main aim of the investigation was to develop measures to prevent the development of air-entrainment vortices. Therefore a huge number of model tests with different structures to direct flow, such as groynes, walls and piles, were tested. In particular for case 5a – turbine operation of structure “3-out” - more than 100 series were conducted until the optimal configuration was found! Here, only the initial situation is presented to keep the main purpose of this paper. These difficulties explain without doubt why a numerical way of forecasting vortices formation is strongly needed.

In case 5, 5a and 6b a large air-entraining vortex has been observed in front of the outlet “3-out”. It has developed because water mainly flowed at the left side of the building around it and then coming back from the right side.

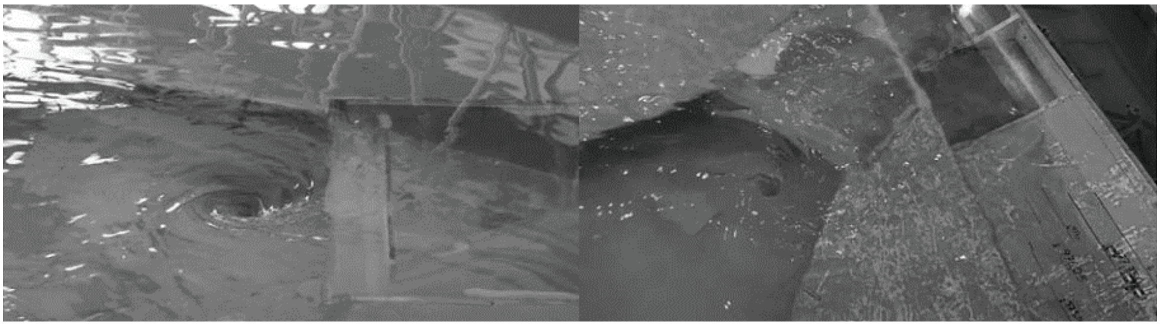


Figure 3 : Air entraining vortex at 3-out (left case 5 and right case 6b

4 NUMERICAL MODEL

Only the Case 5 has been calculated to assess the method

- It means that the initial water level is 978.5 m, the final water level is 975.5 m.
- The flow rate in the inlet (Q2) is 80 m3/s
- The flow rate in the outlet(Q3) is 110 m3/s

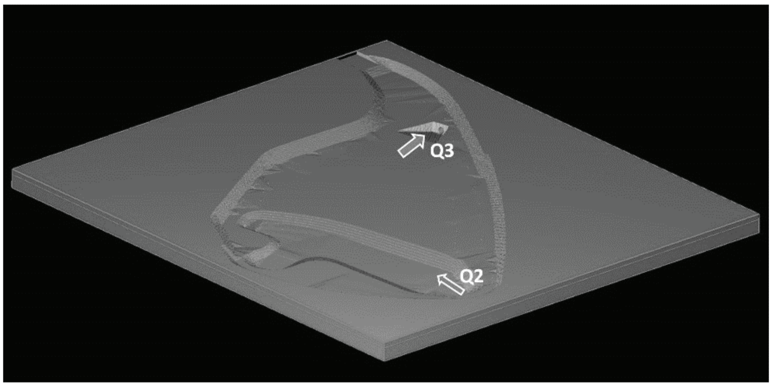


Figure 4 : Numerical model geometry

4.1 *Large domain*

4.1.1 *Mesh*

The first step of this simulation is to simulate the flow behavior everywhere in the reservoir. For that purpose a 0.5 m square mesh has been used. A sensitivity study dealing with the size mesh has proved that wider cell (0.75 m or 1m) may cause free surface instability. The mesh is composed by around 20 million cells.

4.1.2 *Boundary and initial conditions*

Both flow rates (Q2 and Q3) have been imposed to control the volume of fluid inside the domain. The initial level 978.5 m was fixed as initial condition.

4.1.3 *Results*

No results conclusion can be drawn in this first step of simulation. At this step, the recirculation caused by the vortex was indistinguishable to the other recirculations.

4.2 *Intermediate domain*

4.2.1 *Mesh*

The second step of this simulation was to simulate the flow behavior in the half of the reservoir to improve the accuracy of the flow patterns in the basin. For that purpose a 0.25 m square mesh has been used. The mesh also was composed by around 20 million cells.

4.2.2 *Boundary conditions*

Q3 flow rate was imposed to control the volume of fluid inside the domain. The results of the previous simulation were fixed as upstream boundary condition.

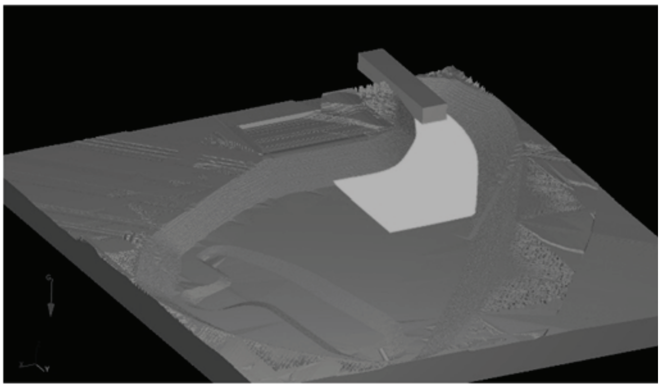


Figure 5 : View in soft grey of the intermediate calculation domain

4.2.3 *Results*

On the following figure for a 977.9 m water level in the basin, it can be noticed that a strong vortex may appear in the zone where Q criterion is high on the right bank upstream the outlet. The same phenomenon has been observed with different water levels in the reservoir.

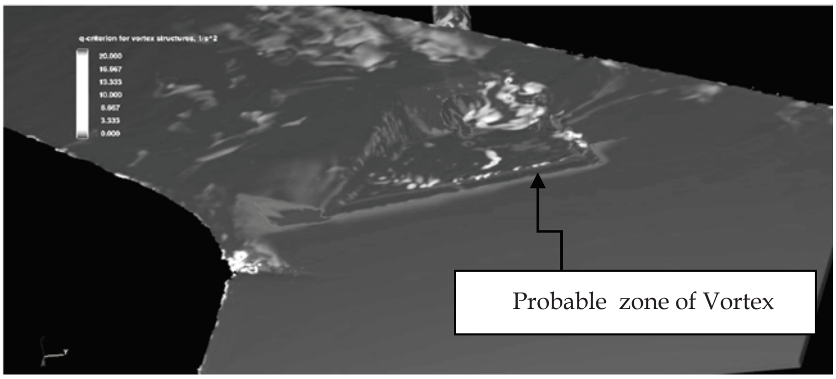


Figure 6 : View of the Q criteria value in the intermediate domain

4.3 *Small domain*

4.3.1 *Mesh*

The third step of this simulation was to simulate the flow behavior close to the outlet where a vortex may appear. For that purpose, a 0.05 m square mesh has been used. The mesh was also composed by around 22 million cells. It must be underlined that the mesh is a full cubic mesh as shown on the following picture.

As studied in the [20], if the mesh is not cubic, the vortex shape may be heavily warped.

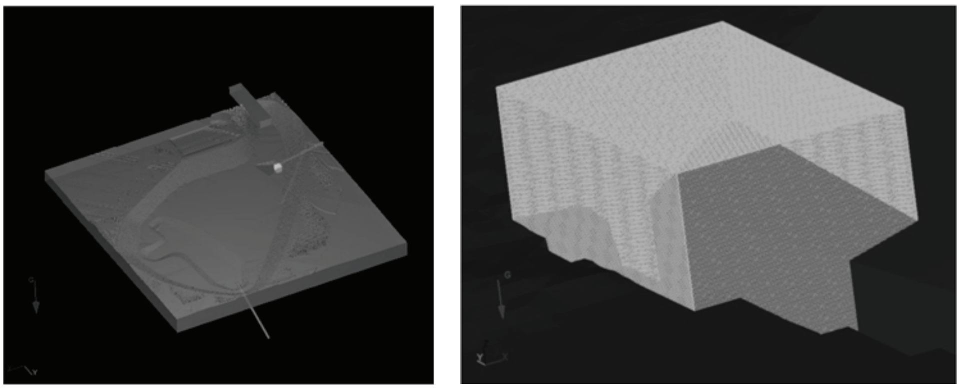


Figure 7 : Plan view of the small calculation domain the calculation domain is the little cube in soft grey and zoom of the mesh

4.3.2 *Boundary* conditions

The results of the previous simulation were fixed as boundary condition.

4.3.3 *Results*

On the following figure for a 977.9 m water level in the basin, it can be noticed that a strong vortex in the zone which has been determined in the previous step occurs. A vortex has been observed when the reservoir level is emptying from the 978.5 to 975.5 m. The vortex clearly appeared at the level of 978.0 m. As shown on the Figure 3 on the physical model the vortex is also clockwise.

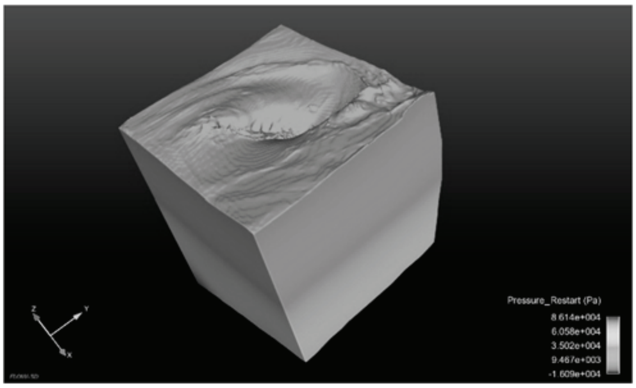


Figure 8 : View of vortex colored by the pressure value in the small domain

Nevertheless, it is still impossible to build a finer mesh which enables to predict the right shape of the vortex. Our current computational power is too poor.

For practical reasons, only a qualitative comparison between the two models has been achieved. It would be undoubtedly worthwhile to pursue with a comparison of the velocity fields near the water intake or the streamlines patterns, but in this case no measurements of the velocities in the physical model are available. However, the method presented herein easily enables to forecast that a risky vortex may appear. This method is based upon the fact that the tested configuration has to be modified if no vortex can be accepted. Thanks to this precious tool it is possible to optimize the design steps and to propose an accurate design solution as the initial configuration of an indispensable scale model. Consequently it may enable to quicken the scale model tests and save invaluable time.

5 CONCLUSIONS

A simplified method to predict the risk of free surface vortices at intakes with 3D CFD Code named Flow 3D has been qualitatively compared to the results of a large scale model in this paper.

Even if the configuration of the selected scale is fairly complex, the numerical simulation was able to detect the risk of vortices.

For that purpose, the method is based on a VOF method with a fine mesh resolution and a high momentum advection order. It also is important to point out the fact that a full cubic mesh really is needed to avoid any doubt concerning a vortex disturbance.

In those conditions only, the submitted method is able to answer to the questions:

- Is there a risk of vortex generation in this configuration?
- Where may a vortex be located?

But it is still impossible to predict the intensity of the phenomenon.

Finally, this tool can be used for engineering processes to design hydraulic structures in the first steps of a project or to optimize a physical model program. However, the intensity of the vortices still should be observed on a scale model.

Nevertheless, the qualitative comparison may be transformed to a quantitative comparison to become even more relevant if a dedicated experimental case can be found.

Although it is emphasized that numerical models must be carefully handled. Hence to assure the reliability of the results it is essential to perform sensitivity tests on the main parameters.

REFERENCES

[1]. Burgers J.M.,(1948). A mathematical model illustrating the theory of turbulence, Adv in Applied Mechanics, 1, 171-199.

[2]. Gordon, (1970). Vortices at intakes, International Water Power & Dam Construction, No 4, p. 137-138.

[3]. Knauss J., (1972), Wirbel an Einläufen zu Wasserfraftanlagen, Technical report, Versuchtanstal für Wasserbau der Technischen Hochchule München.

[4]. Jain A. K., Ranga raju K. G., Garde R. J., (1978). Vortex formation at vertical pipe intakes. Journal of the Hydraulics Division, ASCE, volume 104, n°10, pp1429-1445.

[5]. De Siervi F., Viguier H. C., Greitzer E. M., Stan C. S. ,(1982). Mechanisms of inlet-vortex formation, Journal of Fluid Mechanic, volume 124, pp. 173-207.

[6]. Knauss J, (1987). Swirling flow problems at intakes, Hydraulic structures design manual, IAHR

[7]. Padmanabhan M., Hecker G. E., (1984). Scale effects in pump sump models. Journal of Hydraulic Engineering, volume 110, n°11, pp1540-1556.

[8]. Levi, E. (1991). Vortices in Hydraulics. Journal of Hydraulics Engineering, volume 117, n°4, pp.399-413.

[9]. Chong M. S., Perry A. E., Cantwell B. J. ,(1990). A general classification of three-dimensional flow fields. Journal "Physics of Fluids", A 2 (5).

[10]. Levy Y., Degani D., Seginer A., (1990). Graphical visualization of vertical flows by means of helicity. AIAA Journal, volume 28, n°28.

[11]. Jeong J., Hussain F. ,1995. On the identification of a vortex. Journal Fluid Mechanic, vol. 285, pp. 69-94.

[12]. Hunt J. C. R., Wray A. A., Moin P, (1998). Eddies,stream, and convergence zones in turbulent flows. Centre for Turbulence Research Report CTR-S88, p193.

[13]. Zhou, Adrian, Balachandar, Kendall, (1999). Mechanisms for generating coherent packets of hairpin vortices. Journal of Fluid Mechanics, 387, p. 353-396.

[14]. ANSI, (1998). Pump Intake Design. American National Standard Institute ANSI/HI 9.8-1998.

[15]. Tokyay T. E., Constantinescu S. G., (2005). Large eddy simulation and Reynolds averaged Navier-Stokes simulations of flow in a realistic pump intake: a validation study. Journal of ASCE.

[16]. Minisci E., Telib H., Cicatelli G., (2005). Hydraulic design validation of the suction intake of a vertical centrifugal pump station, by use of computional fluid dynamic (CFD) analysis. Proceeding of the ASME Fluids Engineering Division Summer Meeting and Exhibition, june 19-23 2005, Houston, USA.

[17]. Hager W., SChleiss A., traité de Génie Civil, seconde édition, (2009) : Constructions hydrauliques, TGC volume 15, Génie Civil 3ème année, Presses polytechniques et universitaires romandes.

[18]. Guyot G., Maaloul H., Archer A., (2012,) A Vortex modeling with 3D CFD, congress Simhydro 2012

[19]. Suerich-Gelick F., Gaskin S., Villeneuve E, Parkinson, (2013) Engineering Approach to Predicting Free Surface Vortice at Low-Head Hydropower Intakes, 2013, IAHR world congress 2013

[20]. Guyot G., PITTION-ROSSILLON A.., Archer A.,(2014) An attempt to propose a simplified method to predict risk of vortices based on a 3D CFD Model compared with an experimental Case, 2014 IAHR European congress

[21]. Huber B., Sollerer F.,(2014) Prevention of air-entrainment at the outlet of a pumped storage plant – large scale model tests, 2014 IAHR European congress