EXPERIMENTAL VERSUS NUMERICAL RESULTS ON THE VELOCITY FIELD IN THE WAKE OF A HYDROPOWER FARM EQUIPPED WITH THREE ACHARD TURBINES

Andrei-Mugur GEORGESCU[1], Sanda-Carmen GEORGESCU[2], Costin Ioan COŞOIU[3], Nicolae ALBOIU[4], Ana-Maria PETRE[5]

Acest studiu tratează curgerea apei în siajul unei ferme hidroelectrice echipate cu 3 module de turbină Achard. Turbina Achard este un concept franţuzesc de turbină hidraulică transversală cu ax vertical, cu trei pale verticale de tip delta, destinată funcţionării în curenţi marini. S-au comparat rezultatele experimentale şi numerice 2D aferente distribuţiei de viteze în aval de 3 turbine Achard, aliniate pe acelaşi rând, transversal pe direcţia de curgere a apei. Câmpul de viteze în aval de fermă s-a măsurat cu un anemometru acustic cu efect Doppler. Simularea numerică 2D a curgerii nestaţionare într-o astfel de fermă s-a realizat atât cu FLUENT (cu model de turbulenţă $k-\omega$ SST), cât şi cu COMSOL Multiphysics (cu model de turbulenţă $k-\varepsilon$). S-au obţinut concluzii privind acurateţea modelării 2D.

This study focuses on water flow in the wake of a hydropower farm equipped with 3 Achard turbine modules. The Achard turbine is a French concept of vertical axis, cross-flow, marine current turbine module, with three vertical delta blades. The comparison between experimental and 2D numerical results has been performed for the velocity distribution downstream of 3 Achard turbines, aligned on the same row across the water flow. The velocity field in the wake of the farm has been measured by Acoustic Doppler Velocimetry. The 2D modeling of the unsteady flow inside such a farm has been performed with both FLUENT (with $k-\omega$ SST turbulence model) and COMSOL Multiphysics (with $k-\varepsilon$ turbulence model). Some conclusions on the accuracy of the 2D modeling were derived from this study.

Keywords: Achard turbine, cross-flow current turbine, power coefficient.

1. Introduction

The Achard turbine is a new concept of vertical axis, cross-flow, marine current turbine module [1], developed since 2001 at LEGI (Geophysical and

[1] Associate Prof., Hydraulics and Environmental Protection (HEP) Department, Technical University of Civil Engineering Bucharest, Romania, andreig@mail.utcb.ro
[2] Associate Prof., Hydraulics, Hydraulic Machinery and Environmental Engineering Department, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania
[3] Lecturer, HEP Department, Technical University of Civil Engineering Bucharest
[4] Lecturer, HEP Department, Technical University of Civil Engineering Bucharest
[5] Student, Building Services Faculty, Technical University of Civil Engineering Bucharest

Industrial Fluid Flows Laboratory) of Grenoble, within the French HARVEST Project. The Technical University of Civil Engineering Bucharest (UTCB), together with the University “Politehnica” of Bucharest, and with the Romanian Academy – Timi□oara Branch, studied the hydrodynamics of Achard turbine modules, within the Romanian THARVEST Project [2].

The main advantages of Achard turbines are their modularity, and their ability to operate irrespective of the water flow direction. Thus, similar modules can be superposed to form towers, with lengths adapted to current depths. A marine or river power farm consists of a cluster of barges, each barge gathering several parallel rows of towers that can be put in non-overlapped, or in overlapped staggered arrangements. For the former case, downstream turbines are not placed in the wake of upstream turbines.

The optimum spatial arrangement of the towers in the farm corresponds to the best overall efficiency. To be able to assess this optimal arrangement in the simplest way possible, i.e. by 2D numerical modeling, we had to make sure that our numerical model was accurate with respect to experimental results. Hence, a simple 1:5 scale model of a power farm equipped with three Achard turbines (Fig. 1), aligned on the same row across the flow, was built [2] and tested in a water channel at the Hydraulics Laboratory of the Technical University of Civil Engineering Bucharest. Due to the channel depth limitations, the turbines cannot be superposed to form towers.

Fig. 1. Experimental set-up: Hydropower farm model at 1:5 scale (left frame); Variable slope open channel (upper-right frame); Turbine models turning in water (lower-right frame)

2. Achard turbine description

The vertical delta blades of the Achard turbine are shaped with NACA 4518 airfoils, while the radial supports are shaped with straight NACA 0018 airfoils. At full scale, the runner diameter is $D = 1$ m, and the runner height is $H = 1$ m. Along each delta blade, the airfoil mean camber line length varies from 0.18m at mid-height of the turbine, where $z = 0$, to 0.12m at the extremities, where $z = \pm H/2$. The Achard turbine module built at full scale [2], was tested in the aerodynamic tunnel at the Wind Engineering & Aerodynamics Lab. of UTCB.

The vertical axis cross-flow turbines run in stabilized current, so the flow can be assumed to be almost unchanged in horizontal planes along the *z*-axis. This assumption allows performing 2D numerical modeling, for different arrangements of the turbines (towers) within the farm. Vertical axis cross-flow turbines present much more complex flow patterns between the blades than classical axial, free stream turbines. In the former case, for a given horizontal position along the blades and a given velocity U_∞ of the current, the angle of attack α and the relative velocity w change constantly during a complete rotation, defined by the azimuthal angle θ from 0^o to 360^o. As a consequence, the lift and drag forces acting on the blades change also during a complete rotation. Those forces can be decomposed with respect to the rotation circle, as normal force F_n and tangential force F_t. The resulting total tangential force is the one responsible for turbine rotation. This is just a theoretical case, where it has been assumed that the blade is subjected to a constant upstream velocity distribution, independent of its position. This is a good assumption for any position of the blade in the first half rotation, for $\theta = 0^\circ \div 180^\circ$. But for $\theta = 180^\circ \div 360^\circ$, due to dynamic stall, the blades are subjected to a different velocity profile. Accordingly, for different time moments during the second half rotation of the turbine, the resulting forces on the blade are different [3], with respect to the theoretical case presented above.

3. Experimental setup

The experimental tests were performed in the variable slope, open channel of the Hydraulics Laboratory of UTCB (Figure 1). That water channel has the following dimensions: 1.2m wide, 0.8m deep, and 28m long. The farm model has been placed at mid length of the channel. In order to achieve the desired experimental flow conditions, the channel slope was set at 1‰; thus, a 0.53m/s mean velocity for a 0.36m water depth was ensured. The turbine models were built at 1:5 geometric scale, resulting with 0.2m diameter and 0.2m height (Fig.1).

Using the similitude conditions derived for such vertical axis turbines [4], the computed value of the rotational speed, which would assure the same tip speed ratio as the real module, is 101rpm. The turbine models are driven by electrical motors provided with variable speed, to insure the computed rotational speed.

Multiple experiments were performed for three turbine disposals as shown in Figure 2. The experiments were performed in two steps: the first one focused on the calibration of the channel and the second one on effective measurements of the velocity distribution downstream of the turbines ensemble. The velocity distribution was measured on three components using a 3D Acoustic Doppler Velocimeter, having a 25Hz sampling rate. The probe samples a 3÷9mm long water volume, with a 6mm diameter. The signal emitted by probe’s transducer is digitally processed by an acquisition board, on a PC. Using the software provided by probe’s manufacturer, the mean velocity components along the channel were obtained. The considered sampling points were located two and a half turbine diameters downstream of the last turbine reached by the flow, on a 0.64m width centered to the channel longitudinal axis. The sampling density was set at 1 measurement/centimeter. In this paper we will present only the results obtained for the single row disposal of the turbines in the channel (Figure 2a).

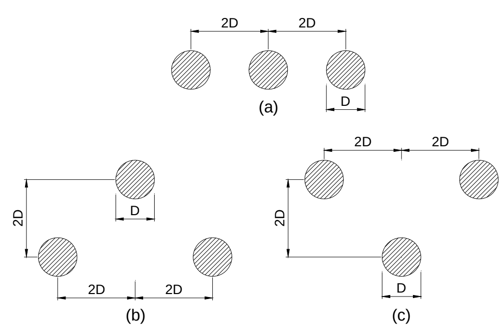

Fig. 2. Disposal schemes of the turbine models: (a) Single row disposal, across flow direction; (b) Two rows disposal with a single front turbine; (c) Two rows disposal with two front turbines

4. Numerical setup in FLUENT

In FLUENT, the simulation was performed for a two-dimensional fluid domain, included in the horizontal plane that intersects the turbines at a distance equal to 0.25 turbine diameters, relative to the median turbine plane (Figure 3). Simulation domain is built to 1:1 scale of the experimental model; its width of 1.2m is equal to that of the channel where the experimental tests were conducted. Horizontal distance between the input section in the field and the axes of rotation of the turbines is equal to 4 diameters of the turbine, i.e. 0.8m. The distance down-stream from the vertical plane containing the axes of rotation of the turbine is equal to 1.4m, i.e. 7 diameters of the turbine. To capture the effects of rotating blades, a sliding mesh model (SMM) was used. Thus, circular areas centered on the axes of rotation of the 3 turbines, with a diameter of 0.24m, were considered to rotate with angular velocity corresponding to a tip speed ratio $\lambda = 2$.

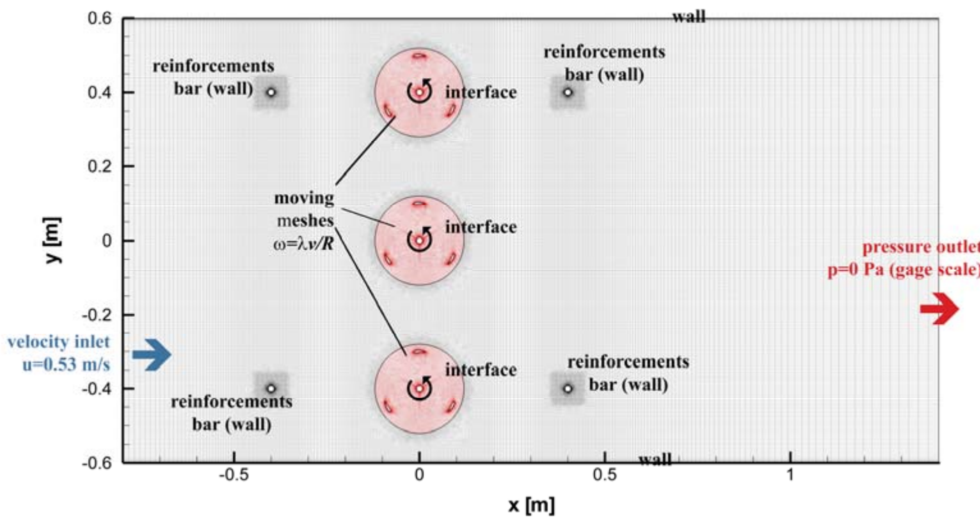

Fig. 3. Grid and boundary conditions used in FLUENT

Table 1

Parameters of FLUENT simulation

Parameter name	Used
Spatial model	2D
Turbulence model	$k-\omega$ SST
Flow type	unsteady
Solver	pressure based, coupled
Discretisation schemes	second order
Solver precision	double precision

The computing grid is a mixed one, made from quad type cells, smaller near the solid surfaces and larger in the free stream zone. The total number of cells is equal to 216497, with a minimum characteristic size of 0.1mm, in the boundary layer areas, adjacent to solid surfaces. At the inlet section, a uniform velocity distribution was considered, with a magnitude of 0.53m/s. In the outlet section, the pressure was designated to be equal to 0, gage scale. On all solid surfaces, a no-slip condition was considered. Rotating fluid domains moved with an angular velocity $\omega = 10.6\,s^{-1}$, corresponding to $\lambda = 2$. The parameters used in the simulation are summarized in Table 1. Around the turbine, the flow regime is turbulent, characterized by parameters that fluctuate strongly in time. Therefore, to capture as accurately as possible the velocity and pressure fields in the considered computational domain, a $k-\omega$ SST turbulent model was adopted. The chosen solver was a double precision, pressure-based coupled one. To obtain a good discretisation of the momentum, continuity and turbulence model specific equations, second-order discretisation schemes were used.

5. Numerical setup in COMSOL Multiphysics

In COMSOL Multiphysics, the simulation was performed for a two-dimensional fluid domain, in the horizontal plane that intersects the turbine at a distance equal to 0.25 turbine diameters, relative to the median turbine plane. The numerical model of the turbine was built at 1:1 geometrical scale as the real Achard turbine, thus the open channel was set to be 5 times bigger (i.e. 6m width).

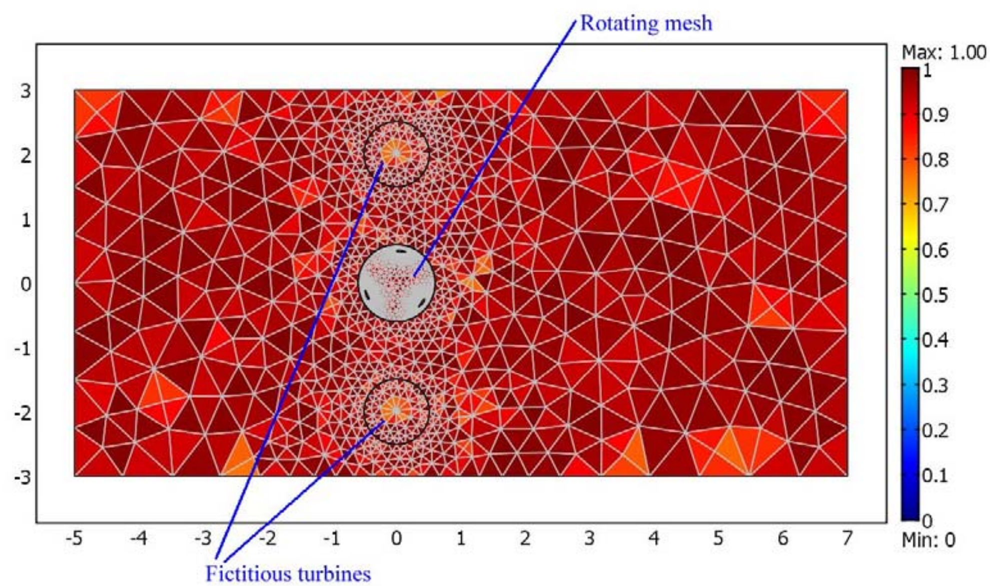

Fig. 4. Mesh quality and numerical setup used in COMSOL Multiphysics

The computational domain was set to 12 turbine diameters long and 6 turbine diameters wide. The resulting mesh consisted of about 5016 triangular elements and 426 boundary elements, yielding a total of 44543 degrees of freedom (Figure 4). The simulation was performed under a time dependent flow regime with a time step of 0.05s. The flow was considered turbulent, with a $k-\varepsilon$ turbulence model. The boundary conditions used were: water inlet with the velocity of 0.53m/s (on the left hand side of Figure 4); water outlet with no viscous stress (on the right hand side of Figure 4); rough wall computed with the logarithmic wall function with an offset of $h/2$ on the top and bottom of the domain, as well as on the blades of the turbine; all other boundaries were set to neutral. In order to gain computational time, we used only one rotating turbine, the two other turbines being modelled based on the action-reaction principle from a single rotating turbine model [5]. The procedure used to replace the forces exerted by the blades on the flow has been exhaustively explained in [6].

6. Comparison between experimental and numerical results

The numerical results and measured experimental data for the non-dimensional velocity profile (as ratio between the velocity and the mean velocity), at 2.5 turbine diameters downstream after the turbines, are presented in Fig. 5.

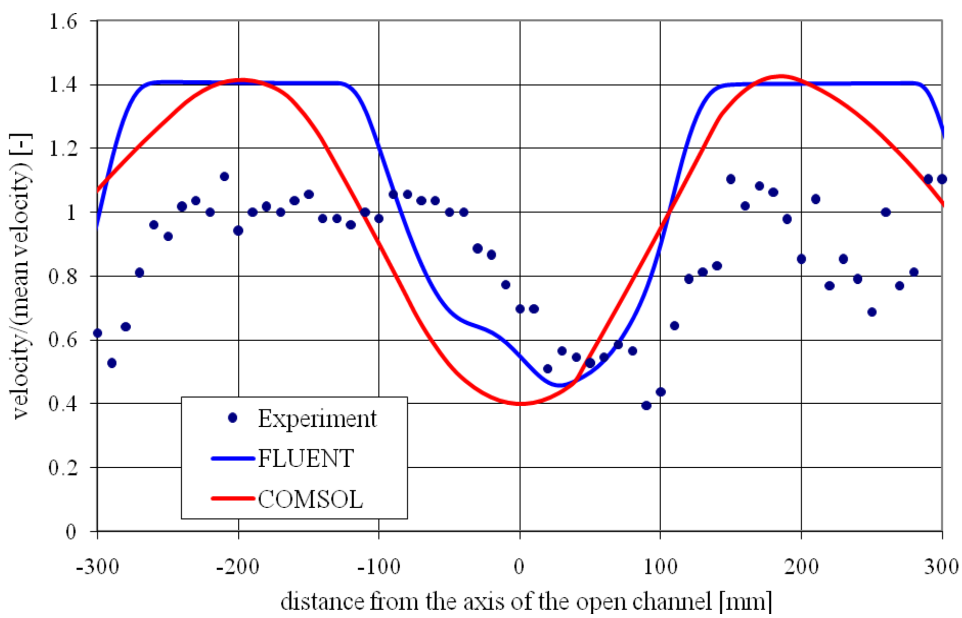

Fig. 5. Comparison between numerical and experimental results of non-dimensional velocity distributions obtained at 2.5 turbine diameters downstream of the turbines alignment

From Fig. 5 we can see that the trends of the curves are somehow similar. The numerical values seem to be closer to one another with respect to the experimental values. Moreover, velocity values between the turbines are greater for the numerical models. This discrepancy is probably due to the fact that for the numerical simulations we used in both cases a 2D model. In fact for the experimental model, water can by-pass the turbines, by flowing above or below, not only sideways as in the numerical simulations.

7. Conclusions

The comparison between experimental and numerical results has been performed for the velocity distribution downstream of three Achard turbines, aligned on the same row across the flow, within a hydropower farm model.

The differences between the numerical values can have two causes. On the first hand, the $k-\omega$ turbulence model used in FLUENT seems to be better suited for this type of problems, than the $k-\varepsilon$ turbulence model used in COMSOL Multiphysics. On the other hand, the global action-reaction model used in COMSOL Multiphysics for 2 out of the 3 turbines is an approximation. It yields of course different results between the turbines. In the COMSOL case, we should only compare the results obtained in the wake of the rotating turbine (the one in the middle of the domain), i.e. between -100mm and 100mm in Figure 5. In that zone, the results are not extremely different. It should be also noted that the COMSOL results curve is not centred on the axis of the turbine, as it may seem from the graph (see Figure 5): the minimal velocity is a little bit shifted to the right of the axis. This can be also due to the sparser grid we used in COMSOL, to gain computational time. More experimental work is to be performed in other configurations of the turbines, to be able to accurately draw conclusions on the usefulness and limitations of 2D numerical models for this type of problems.

R E F E R E N C E S

[1]. *J.-L. Achard and T. Maître*, Turbomachine hydraulique. Brevet déposé, Institut National Polytechnique de Grenoble, France, Code FR 04 50209, 2004

[2]. *A.-M. Georgescu, Sanda-Carmen Georgescu, S. I. Bernad et al*, Inter-influence of the vertical axis, stabilised, Achard type hydraulic turbines – THARVEST, CEEX Project No. 192/2006, Bucharest, 2006-2008, http://www.tharvest.ro

[3]. *Sanda-Carmen Georgescu, A.-M. Georgescu, S. I. Bernad and R. Susan-Resiga*, "Overall efficiency of hydropower farms consisting of multiple vertical axis, cross-flow, marine current turbine modules", in Proc. CMFF'09, Budapest, ed. J. Vad, **II**, 2009, pp. 752-759

[4]. *A.-M. Georgescu, Sanda-Carmen Georgescu, M. Degeratu, S. Bernad and C. I. Coşoiu*, "Numerical modelling comparison between airflow and water flow within the Achard-type turbine", in Proc. 2nd IAHR WG CDPHMS, Timi□oara, Romania, 2007, pp. 289-298

[5]. *Sanda-Carmen Georgescu, A.-M. Georgescu and S. I. Bernad*, "Two-dimensional simulation of the unsteady flow through the Achard turbine: COMSOL Multiphysics versus Fluent results", in U. P. B. Sci. Bull., Series C, **vol. 69**, no. 4, 2007, pp. 635-642

[6]. *A.-M. Georgescu, Sanda-Carmen Georgescu and S. I. Bernad*, "A method to assess the inter-influence between vertical axis, cross-flow turbines in a free stream – 2D numerical modelling", in Proc. COMSOL Conference 2008, Budapest, Hungary, 2008, pp. 45-51.