TWO-DIMENSIONAL SIMULATION OF THE UNSTEADY FLOW THROUGH THE ACHARD TURBINE: COMSOL MULTIPHYSICS VERSUS FLUENT RESULTS

Sanda-Carmen GEORGESCU[1], Andrei-Mugur GEORGESCU[2],
Sandor Ianos BERNAD[3]

Two-dimensional numerical modelling of the unsteady flow through the blades of the Achard turbine, a new vertical axis cross-flow water turbine concept, is performed both with COMSOL Multiphysics 3.3a and with Fluent 6.01 software, in order to compare qualitatively the results and the software capabilities. The $k - \varepsilon$ turbulence model has been selected and same geometry and boundary conditions were considered within computations. Global results with respect to the pressure coefficients on the airfoils agree well with experimental data. The 2D computational approaches cannot fully predict the dynamic stall phenomenon that is reported in the case of vertical axis cross-flow turbines. 3D simulations are necessary to obtain an accurate description of the flow around a varying blade cross-section along the z-axis, like the delta blade of the Achard turbine.

Keywords: Achard turbine, cross-flow water turbine, airfoil, delta blade.

1. Achard turbine description

The *THARVEST* Project of the *CEEX* Program sustained by the Romanian Ministry of Education and Research [1-6] aims to study experimentally and numerically the hydrodynamics of a new concept of water-current turbine, called *Achard turbine*, in collaboration with partners from Geophysical and Industrial Fluid Flows Laboratory (LEGI) of Grenoble, involved in the French *HARVEST* Project [7-11]. The Achard turbine, a cross-flow marine or river turbine with vertical axis and delta blades, is studied in France mainly with regard to marine applications, to extract energy from tidal currents in costal locations. But the Achard turbines are also suitable to be placed in big rivers, as the Danube, and to produce the desired power by summing elementary power provided by small turbine modules [12]. In Figure 1 we present the Achard turbine modules piled up

---

[1] Associate Prof., Hydraulics and Hydraulic Machinery Department, University "Politehnica" of Bucharest, Romania
[2] Associate Prof., Hydraulic and Environmental Protection Department, Technical University of Civil Engineering Bucharest, Romania
[3] Senior Researcher, Centre of Advanced Research in Engineering Sciences, Romanian Academy – Timişoara Branch, Romania

in towers [13], within a river hydropower farm, where the turbines can operate efficiently in free flow without dams.
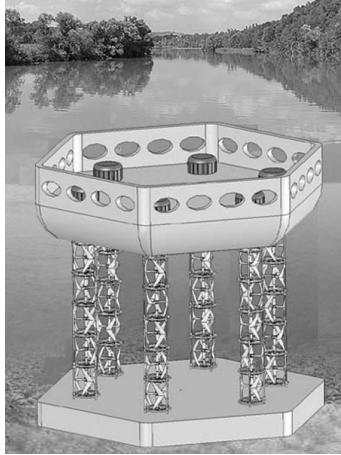

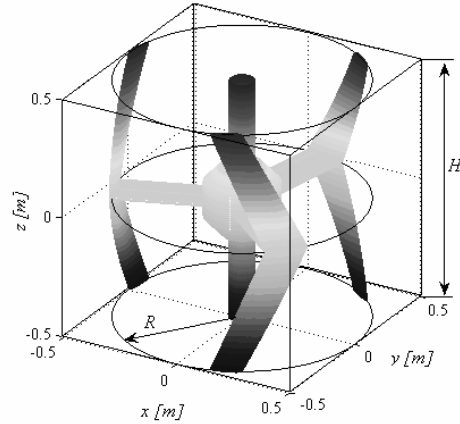Fig. 1. River hydropower farm.

Fig. 2. Achard turbine module.

In Figure 2 we present the 3D geometry of the Achard turbine module, generated fully analytically in MATLAB [2]. This type of turbine consists of a runner with three vertical delta blades, sustained by radial supports at mid-height of the turbine, and stiffened with circular rims at the upper and lower part of the turbine (those rims are not represented in Figure 2, to allow a better view of the runner). The blades are shaped with NACA 4518 airfoils, the radial supports are shaped with straight NACA 0018 airfoils [14], while the circular rims are shaped with lens type airfoil. The turbine main geometric dimensions are: the runner radius $R = 0.5$ m, the runner height $H = 1$ m, and the shaft radius $r_s = 0.05$ m.

The NACA 4518 airfoil corresponding to the turbine blades has the mean camber line along the runner circumference [2], and its maximum thickness, as percent of the chord length, is $d = 18\%$. Along each delta blade, the airfoil mean camber line length c_0 varies from 0.18 m at $z = 0$, to 0.12 m at the extremities, where $z = \pm 0.5$ m. The airfoil chord length c can be expressed as:

$$c = 2R\sin(c_0/2R). \tag{1}$$

Between the leading edge of blade's extremity and the leading edge of the blade at mid-height of the turbine, there is a 30° azimuth angle.

2. Two-dimensional computational domain

The 2D computations correspond to horizontal cross-planes, placed at constant z level values. The values of the azimuth angle of the blades are

$\theta = \left\{0^{o}; 120^{o}; 240^{o}\right\}$, in counter clockwise direction, as in Figure 3. Within this paper, we performed computations for two horizontal cross-planes, namely: one placed at mid-height of the turbine, without radial supports (Fig. 3), where $c_0 = 0.18$ m (the maximum value of c_0 along the delta wing), and the other one placed at $z = 0.25$ m level, where $c_0 = 0.15$ m (the mean camber line length). In Figure 4, we present a zoomed image of the computational domain, including the airfoil profile placed at $\theta = 0^{o}$, for both cases: $c_0 = 0.15$ m, and $c_0 = 0.18$ m.
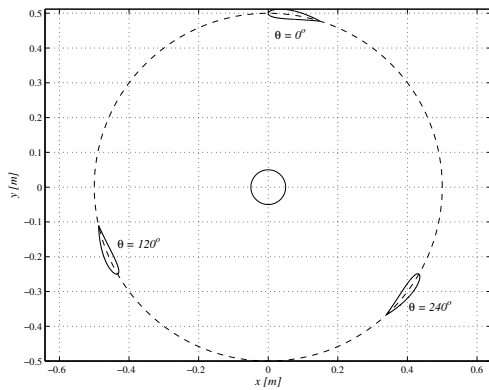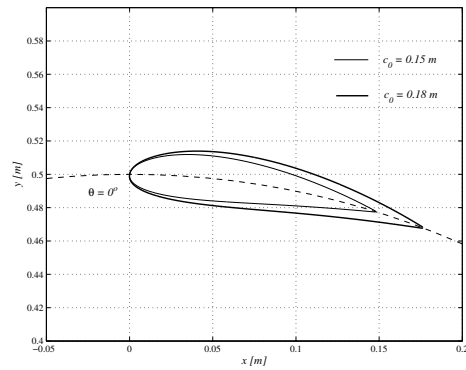

Fig. 3. Runner cross-section for $c_0 = 0.18$ m.

Fig. 4. Airfoil shapes at $\theta = 0^{o}$.

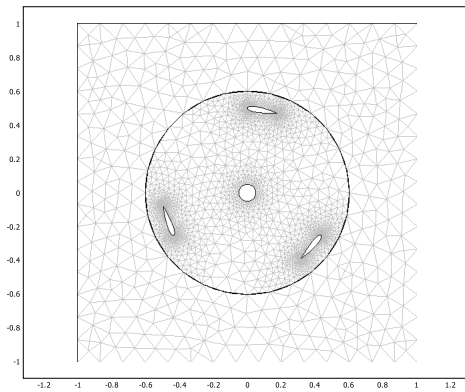
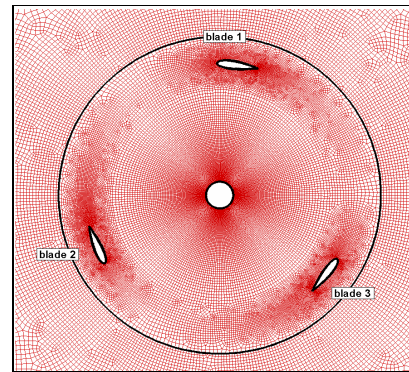Fig. 5. Computational domain discretization in COMSOL Multiphysics ($c_0 = 0.18$ m)

Fig. 6. Computational domain discretization in Fluent ($c_0 = 0.15$ m)

The numerical simulations are performed with COMSOL Multiphysics 3.3a (a Finite Element Method based software), and with Fluent 6.01 (a Finite Volume Method based software), using the $k - \varepsilon$ turbulence model. Both

software use the same geometry of the flow domain. The discretization has the same number of nodes on the profiles, but the total number of cells of the unstructured mesh is slightly different from one software to the other. There are triangular cells in COMSOL (Figure 5), and quadrilateral cells in Fluent (Figure 6). We used two computational sub domains: a rotating one (a circular area of 0.6 m radius that incorporates the blades), and a fixed one (outside the former). The rotating sub domain modelled the rotation of the turbine.

The following boundary conditions are considered: on the left side of the domain (on the water inflow boundary), a constant upstream velocity V_0, a turbulent intensity and turbulence length scale are imposed; on the right side of the domain (on the water outflow boundary), a zero relative pressure is considered; on the upper wall, as well as on the lower wall of the domain, a slip symmetry condition is selected; on the airfoil surface and on the turbine shaft, a logarithmic wall function is selected.

3. Numerical results

Within this section, we present some of the results obtained for the 2D numerical modelling of the unsteady flow through the Achard turbine. Qualitative comparisons between COMSOL Multiphysics and Fluent results are performed for the velocity field, vorticity field, and turbulent kinetic energy field. The displayed results obtained in COMSOL correspond to $c_0 = 0.18$ m and $V_0 = 1$ m/s. The displayed results obtained in Fluent correspond to $c_0 = 0.15$ m and $V_0 = 4.71$ m/s, as in Bernad et al [3]. The results show that the flow behaviour description is similar in both COMSOL Multiphysics and Fluent software. Unfortunately, it is difficult to fit the same grey-colour scheme within both software post-processors.

To illustrate the flow structure obtained for the numerical simulations performed with COMSOL Multiphysics, we present in Figure 7 the evolution of the velocity field, in Figure 8 the evolution of the turbulent kinetic energy field (in logarithmic values), and in Figure 9 the evolution of the vorticity field, at different azimuthal angles. The values $\theta = \left\{ 60^{\mathrm{o}}; 120^{\mathrm{o}}; 150^{\mathrm{o}}; 180^{\mathrm{o}}; 210^{\mathrm{o}}; 300^{\mathrm{o}} \right\}$ correspond to the position of the first blade during its rotation, that first blade being the one placed initially at $\theta = 0^{\mathrm{o}}$ in Figure 3. As the 3 blades are separated by 120°, the above selected values of θ cover the whole complete turn. The flow structure obtained with Fluent is presented in Figure 10, where the evolution of the velocity field is plotted in the upper frames, while the evolution of the vorticity field is plotted in the lower frames. The selected azimuthal angles are $\theta = \left\{ 60^{\mathrm{o}}; 120^{\mathrm{o}} \right\}$, taken with respect to the position of the first blade during its rotation.
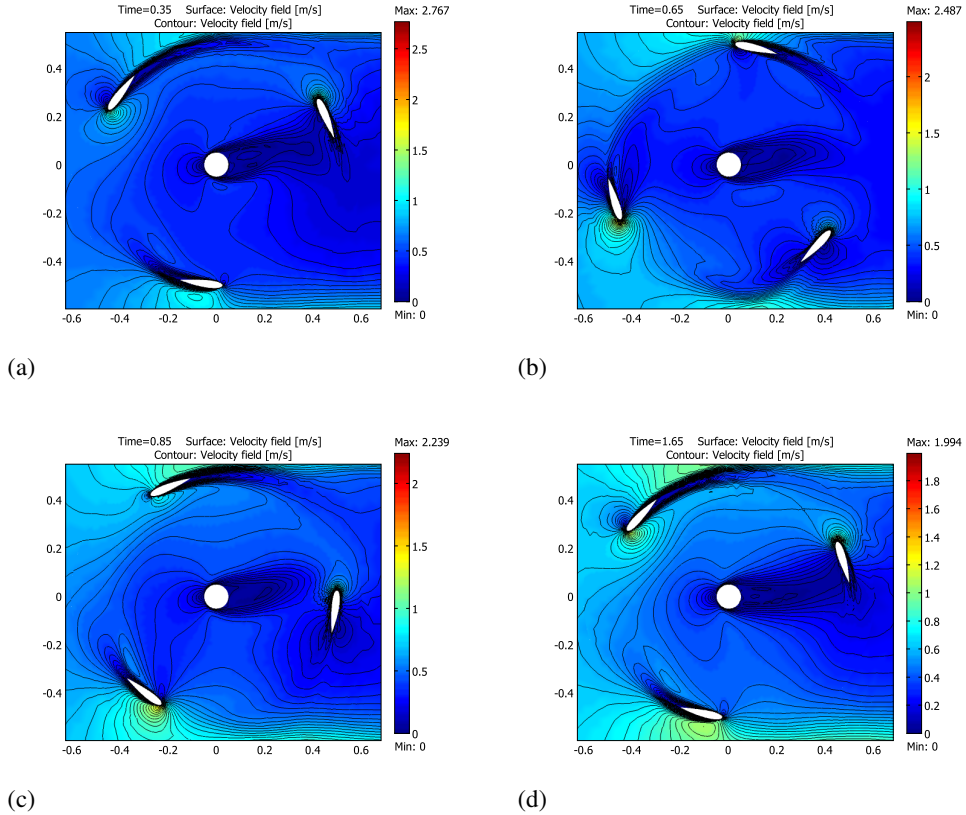
Fig. 7. Velocity field in COMSOL: (a) $\theta = 60^o$, (b) $\theta = 120^o$, (c) $\theta = 150^o$ and (d) $\theta = 300^o$

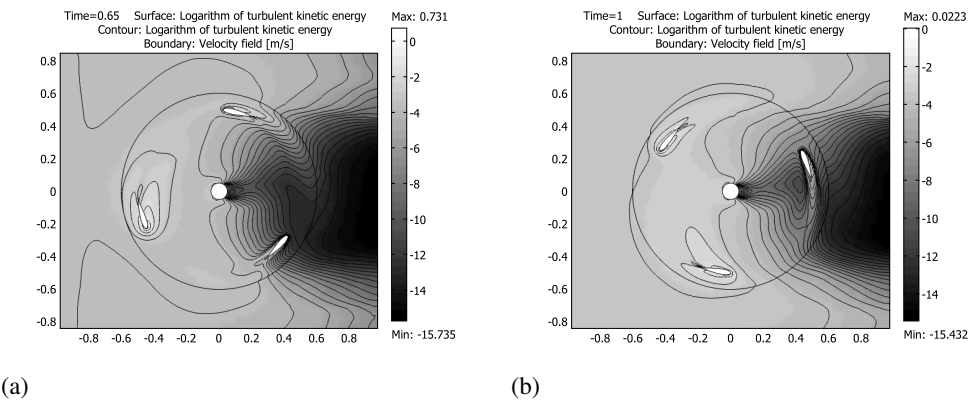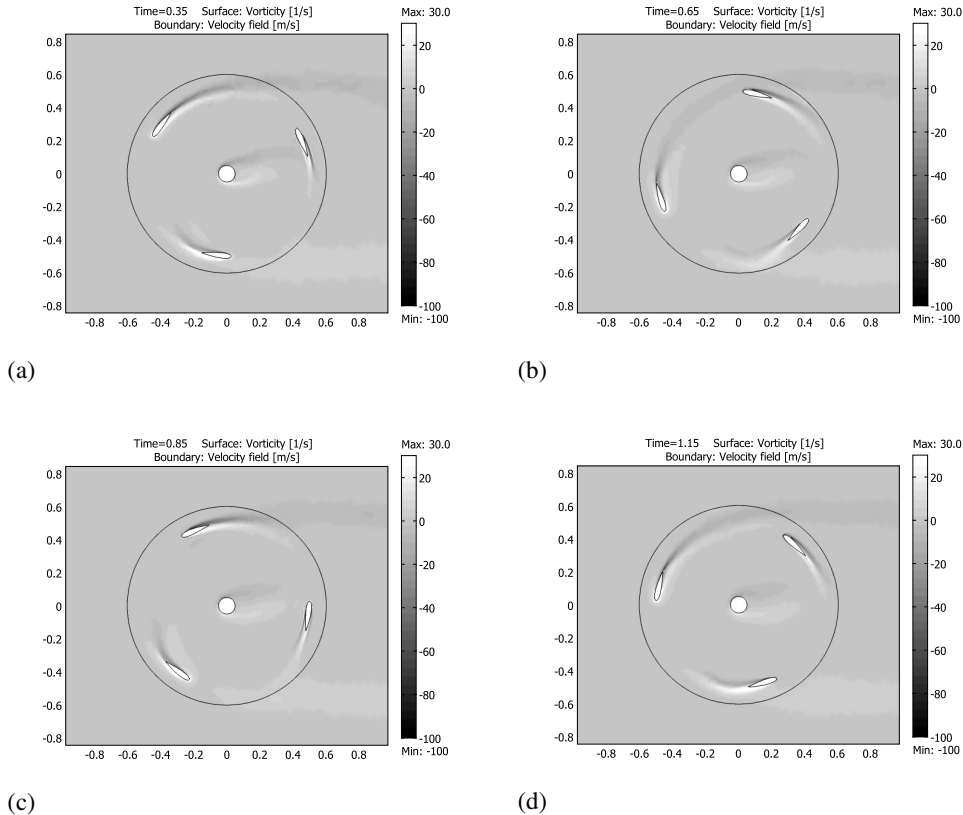Fig. 8. Logarithm of turbulent kinetic energy in COMSOL: (a) $\theta = 120^o$ and (b) $\theta = 180^o$

(a)

(b)

(c)

(d)

Fig. 9. Voticity field in COMSOL: (a) $\theta = 60^{\text{o}}$, (b) $\theta = 120^{\text{o}}$, (c) $\theta = 150^{\text{o}}$ and (d) $\theta = 210^{\text{o}}$

Global numerical results with respect to the pressure coefficients on the blades agree well with available experimental data [2, 5].

4. Conclusions

Within this paper, 2D numerical computations are performed both with COMSOL Multiphysics 3.3a software and Fluent 6.01 software, in order to depict the unsteady flow behaviour through the blades of the Achard turbine. Both software produce similar results.

By analysing the unsteady flow in horizontal cross-sections of the turbine, at different z level values, we can obtain an acceptable description of the various field functions. Unfortunately, the 2D computational approaches cannot fully predict the dynamic stall phenomenon that is reported in the case of vertical axis cross-flow turbines. Some field function variations along the z-axis need to be

considered, so 3D simulations are necessary to obtain an accurate description of the flow around a varying blade cross-section along the z-axis, like the delta blade of the Achard turbine. Since COMSOL Multiphysics is limited to 2D simulations of the fluid flow in hydraulic machineries, 3D simulations for the unsteady flow in the Achard turbine will be performed further only with Fluent.
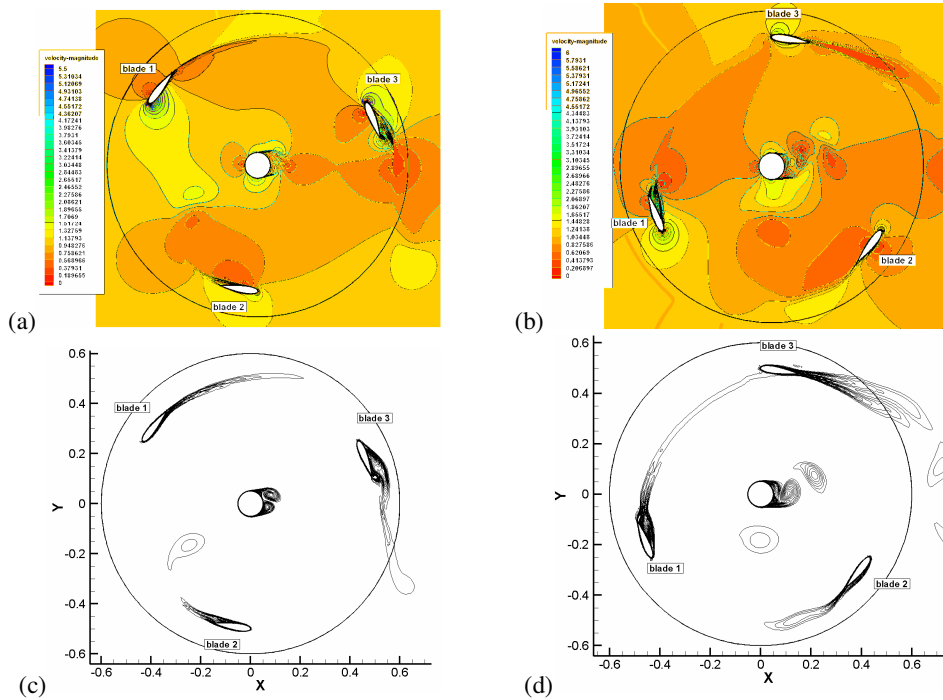

Fig. 10. Evolution of the velocity fields (upper frames) and vorticity fields (lower frames), obtained in Fluent: (a) $\theta = 60^{\circ}$ and (b) $\theta = 120^{\circ}$

R E F E R E N C E S

[1]. *A.-M. Georgescu, Sanda-Carmen Georgescu, S. I. Bernad et al*., Interinfluence of the vertical axis, stabilised, Achard type hydraulic turbines (THARVEST), CEEX Programme, no. 192/2006, AMCSIT Politehnica Bucharest, http://hidraulica.utcb.ro/tharvest/, 2006-2008.

[2]. *A.-M. Georgescu, Sanda-Carmen Georgescu, S. Bernad and C. I. Coşoiu*, "COMSOL Multi-physics versus Fluent: 2D numerical simulation of the stationary flow around a blade of the Achard turbine", in Scientific Bulletin of the "Politehnica" University of Timişoara, Transactions on Mechanics, **vol. 52**, no. 66, fascicola 3, Special Issue Proc. of the 3rd Workshop on Vortex Dominated Flows (WVDF), June 1-2, eds. S. Bernad, S. Muntean & R. Susan-Resiga, 2007, pp. 13-22.

[3]. *S. Bernad, A.-M. Georgescu, Sanda-Carmen Georgescu, D. Balint and R. Susan-Resiga*, "2D unsteady flow simulation in the Achard turbine", in Sci. Bull. "Politehnica" University of Timisoara, Trans. on Mechanics, **vol. 52**, no. 66, fascicola 3, WVDF, 2007, pp. 23-28.

[4]. *Sanda-Carmen Georgescu, A.-M. Georgescu, R. M. Damian and J.-L. Achard*, "Past and future of water turbines in Romania", in Proc. of the 5th International Water History Association Conference on Pasts and Futures of Water, June 13-17, Tampere, Finland, CD-ROM, 8p., 2007.

[5]. *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 Scientific Bulletin of the "Politehnica" University of Timişoara, Transactions on Mechanics, **vol. 52**, no. 66, Special Issue Proc. of the 2nd IAHR International Meeting of the WorkGroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems (IAHRWG), October 24-26, Timişoara, 10 p., 2007 (to appear).

[6]. *A.-M. Georgescu, Sanda-Carmen Georgescu, S. I. Bernad and Carmen Anca Safta*, **"**COMSOL Multiphysics 2D flow simulation in the Achard turbine", in Lucrările Conferinţei Cercetarea de Excelenţă - premiza favorabilă pentru dezvoltarea spaţiului românesc de cercetare CeEx2007, Braşov, 24-26 Octombrie, 6p., 2007 (to appear).

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

[8]. *J.-L. Achard, D. Imbault and T. Maître*, Dispositif de maintien d'une turbomachine hydraulique. Brevet déposé, Code FR 05 50420, Titulaire: Institut National Polytechnique de Grenoble, 2005.

[9]. *Cristina Ploeşteanu*, Étude hydrodynamique d'une type d'hydraulienne à axe vertical pour les courants marins, Thèse de Doctorat, Institut National Polytechnique de Grenoble, 2004.

[10]. *T. Maître T., J-L. Achard, Lynda Guittet and Cristina Ploeşteanu*, "Marine turbine development: Numerical and experimental investigations", in Sci. Bull. "Politehnica" University of Timişoara, Trans. on Mechanics, **vol. 50**, no. 64, 2005, pp. 59-66.

[11]. *J. Zanette, D. Imbault and A. Tourabi*, "Fluid-Structure interaction and design of water current turbines", in Sci. Bull. "Politehnica" University of Timişoara, Trans. on Mechanics, **vol. 52**, no. 66, IAHRWG, 15 p., 2007 (to appear).

[12]. *S. Antheaume, T. Maître and J.-L. Achard*, "An innovative modelling approach to investigate the efficiency of cross flow water turbine farms", in Sci. Bull. "Politehnica" University of Timişoara, Trans. on Mechanics, **vol. 52**, no. 66, IAHRWG, 9 p., 2007 (to appear).

[13]. *A. Tourabi*, Fluid-Structure interaction application to optimisation of water current turbines (pps file), 3S-R Laboratory, Grenoble, 2007.

[14]. *** UIUC Airfoil Coordinates Database. Version 2.0 (over 1550 airfoils), UIUC Applied Aerodynamics Group, Department of Aerospace Engineering, University of Illinois, Urbana-Champaign, USA, http://www.ae.uiuc.edu/m-selig/ ads/coord_database.html, 2006.