U.P.B. Sci. Bull., Series D, Vol. 74, Iss. 4, 2012 ISSN 1454-2358

TIME EFFICIENT COMPUTING OF THE POWER COEFFICIENT FOR A DUCTED ACHARD TURBINE

Andrei-Mugur GEORGESCU[1], Sanda-Carmen GEORGESCU[2],
Ana-Maria PETRE[3], Ion BADEA[4]

În această lucrare este prezentată o metodă eficientă, din punctul de vedere al timpului de calcul, pentru determinarea coeficientului de putere al unei turbine Achard, plasate într-un carenaj simetric, profilat, într-un domeniu de curgere mărginit. Turbina Achard este o turbină trasversală, cu ax vertical şi trei pale verticale (în delta sau rectilinii), destinată funcţionării în curenţi marini/fluviali. În cazul montării mai multor turbine în turnuri, curgerea este cvasi neschimbată în plane orizontale de-a lungul axei verticale Oz; aceasta permite efectuarea de simulări numerice bidimensionale într-o secţiune transversală printr-o turbină la un anumit nivel z. Modelarea numerică 2D a curgerii nestaţionare în turbina Achard carcasată a fost realizată cu COMSOL Multiphysics. Rezultatele au fost comparate cu rezultate similare obţinute la LEGI Grenoble. Au fost reliefate concluzii asupra acurateţii modelării numerice 2D cu metoda propusă.

We present a time efficient method for computing the power coefficient for a shrouded Achard turbine: a turbine running within a symmetric two-foiled channelling device, in a confined flow domain. The Achard turbine is a vertical axis, cross-flow, marine/ river current turbine, with three vertical blades (of delta shape or straight). When several turbines are mounted in towers, the flow is almost unchanged in horizontal planes along the vertical z-axis; this allows 2D numerical modelling in a cross-section of the turbine at a certain z-level. The 2D modelling of the transient flow inside this turbine was performed with COMSOL Multiphysics. Our results were compared with similar results obtained at LEGI of Grenoble. Conclusions on the accuracy of the proposed 2D modelling method were derived.

Keywords: Achard turbine, ducted cross-flow current turbine, channelling device, power coefficient, blockage coefficient

1. Description of the ducted Achard turbine

By analogy to wind turbines, one can extract energy from marine/river currents with almost zero environmental impact, by converting the kinetic energy of flowing water directly to mechanical power, without interrupting the natural flow [1]. In contrast to wind, water currents can be accurately predicted. The

[1] Associate Prof., Hydraulics and Environmental Protection Department, Technical University of Civil Engineering Bucharest, Romania, e-mail: andreig@hidraulica.utcb.ro
[2] Associate Prof., Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania
[3] MSc, Building Services Faculty, UTCB
[4] BSc Student, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania

upper theoretical limit of the power coefficient of an isolated current turbine in open flow equals:

- 16/27 or 0.593 for the axial-flow type turbines, which is the so-called Betz limit, defining an upper limit of 59.3% of the incident kinetic energy that can be extracted from the fluid for a single actuator disk (surface across which energy is extracted as the flow passes through it);
- 16/25 or 0.64 for the cross-flow type turbines [2], defining an upper limit of 64% of the incident kinetic energy that can be extracted from the fluid for a double actuator disk (such as Darrieus, Gorlov and Achard cross-flow turbines).

In open flow, the reported efficiency (power coefficient) of an isolated axial-flow marine current turbine reaches 45% [3], while it is only 20–35% for cross-flow water turbines [4]. To increase their efficiency, water current turbines are equipped with channeling devices, which concentrate the fluid flow through the turbine, thus allowing higher energy extraction levels [5÷7]. For such ducted or “diffuser-augmented” current turbines, the power coefficient increases [5]. For ducted turbines, the power coefficient depends on the pressure drop between duct inlet and outlet, as well as on the flow rate through the duct; these factors depend on duct's shape and on the ratio of duct area to turbine area.

Another method to increase the efficiency of water current turbines operating in open flow is to build hydropower farms, where several parallel rows of turbines or towers (consisting of several superposed turbines) can be arranged in staggered rows configurations, namely in non-overlapped configurations (where the turbines of the downstream row are not placed in the wake of the upstream turbines), or in overlapped configurations [8]: downstream turbines, which operate in a confined flow domain, have a greater efficiency than the corresponding ones placed in non-overlapped configurations. The optimum spatial arrangement of the turbines in the farm corresponds to the best overall efficiency.

The Achard turbine is a recent French patent [9] of vertical axis, cross-flow, marine/river current turbine, developed in Grenoble since 2001, within the French HARVEST Project (abbreviated from Hydroliennes à Axe de Rotation VErtical STabilisé). The Technical University of Civil Engineering Bucharest (UTCB), together with the University “Politehnica” of Bucharest, and with the Romanian Academy – Timişoara Branch, studied the hydrodynamics of Achard turbines, within the Romanian THARVEST Project [10]. The Achard turbine has three vertical blades (delta blades or straight blades, shaped with NACA 4518 airfoils), sustained by mid-radial supports (shaped with NACA 0018 airfoils), as in figure 1 (frames 1a and 1b). The straight-bladed Achard turbine is the closest concept to straight-bladed Darrieus turbine, the difference being the supports of the blades. For big size Achard turbines, upper and lower disks (figure 1, frame 1c) or rims can be attached to the blades, to reduce fatigue during cyclic loads.

(a) (b) (c)

Fig. 1. Achard turbine with: (a) delta blades; (b) straight blades; (c) delta blades and reinforcing disks (a turbine of 1 m diameter, built in Romania, at UTCB [7]).

Five PhD theses were already finalized at the Laboratory of Geophysical and Industrial Fluid Flows (LEGI) of Grenoble on vertical axis cross-flow current turbines, studied numerically and/or experimentally, namely: isolated turbines in open flow [11÷14], ducted isolated turbines [13, 14], and ducted turbines mounted in towers [15], the last thesis containing also in situ measurements for a prototype operating in the headrace of an EDF hydropower plant near Grenoble. Main results on vertical axis cross-flow current turbines, obtained in the above two projects, were already published, e.g. the last ones in 2010 [16, 17] and 2011 [18].

In the present paper, we propose a time efficient method for computing the power coefficient for an Achard turbine running within a symmetric two-foiled channelling device, in a confined flow domain. When several turbines are mounted in towers, the flow is almost unchanged in horizontal planes along the vertical z-axis; this allows 2D numerical modelling in a cross-section of the turbine at a certain constant z-level. The 2D modelling of the transient flow inside such a turbine was performed with COMSOL Multiphysics, using the $k-\varepsilon$ turbulence model. Our results were compared with similar results obtained at LEGI of Grenoble, by Menchaca Roa [14].

The main dimensions of the Achard turbine are the runner diameter D and the runner height, $H = D$. As mentioned before, the Achard turbine can have delta blades [8÷12, 15, 18] or straight blades [13÷15, 17], shaped with NACA 4518 airfoils [19] (an airfoil with the mean camber line along the runner circumference). Within the cylindrical coordinate system, along each delta blade, the airfoil mean camber line length varies from $c = 0.18D$ at mid-height of the turbine (where $z = 0$), to $c = 0.12D$ at the extremities (where $z = \pm 0.5H$), as in figure 1a. For an Achard turbine with delta blades, the 2D computational plane is usually selected at $z = 0.25H$, where $c = 0.15D$. For the straight-bladed Achard turbine, the airfoil mean camber line length is constant: $c = 0.18D$, along all blade (fig. 1b); the computational plane is taken at $z = 0.25H$.

To compare our results with those of Menchaca Roa [14], in this paper the studied Achard turbine has straight blades, and the following dimensions [13, 14]: runner diameter $D = 0.175\,\text{m}$, shaft diameter $d = 0.022\,\text{m}$, and airfoil mean camber line length $c = 0.18D = 0.032\,\text{m}$.

The above straight-bladed Achard turbine runs within a channelling device, consisting of two symmetric foils that can be mounted at different incidence angles α (figure 2). Two airfoils were already tested [14] for such a channelling device, namely: S1223-RTL and EPPLER 420; data corresponding to those airfoils can be found in AID [20]. In this paper, we used the S1223-RTL airfoil, which geometry is defined at $\alpha = 0^\circ$ by its thickness 13.5%, camber 8.5%, trailing edge angle 10.8º, lower flatness 19.2% and leading edge radius 2.3%; airfoil's normalized coordinates for the unit chord length can be found in AID [20, http://www.worldofkrauss.com/foils/701.dat]. In this paper, the chord length is selected as $L = 2D = 0.35\,\text{m}$ (figure 2b), and the incidence angle has one of the following 6 values: $\alpha \in \{0^\circ; 14^\circ; 18^\circ; 24^\circ; 30^\circ; 34^\circ\}$. Menchaca Roa [14] proved that $\alpha = 30^\circ$ is the optimum incidence angle, being the one attached to the maximum power coefficient. The geometry of the channelling device is defined in figure 2b, namely: $b \cong 0.1\,\text{m}$ (cross-section distance between foil's leading edge and turbine's axis level), $e \cong 0.012\,\text{m}$ (gap between the turbine and each channelling foil); the longitudinal distance a between foil's leading edge and turbine's axis varies for each α value, to fit the imposed gap e.

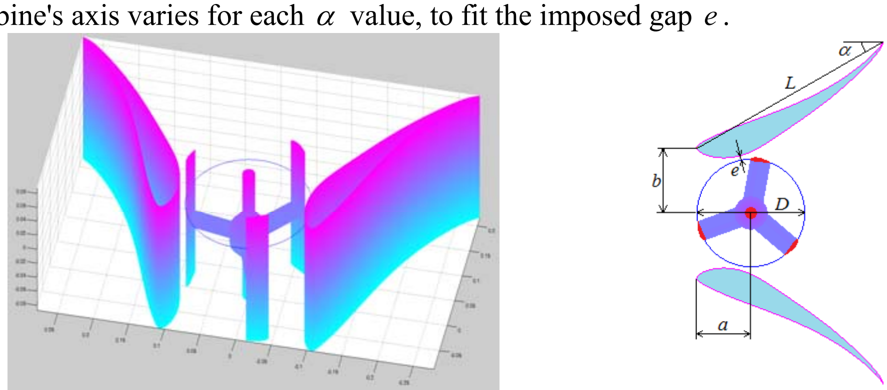

Fig. 2. Ducted Achard turbine: 3D view (left frame) and 2D cross-section (right frame).

2. Computational approach

The 2D numerical model that we created in order to quantify the transient flow inside a ducted Achard turbine was derived by using the COMSOL Multiphysics *Rotating Machinery* approach. All numerical tests were carried out during 12s (representing 6 full rotations of the turbine), with a time step of 0.05s, for a tip speed ratio $\lambda = \omega R / V_\infty \cong 2$, with turbine's radius $R = 0.5D = 0.0875$ m, upstream flow velocity $V_\infty = 2.3$ m/s and turbine's angular velocity $\omega = 52.36$ rad/s (meaning a rotational speed of 500rpm).

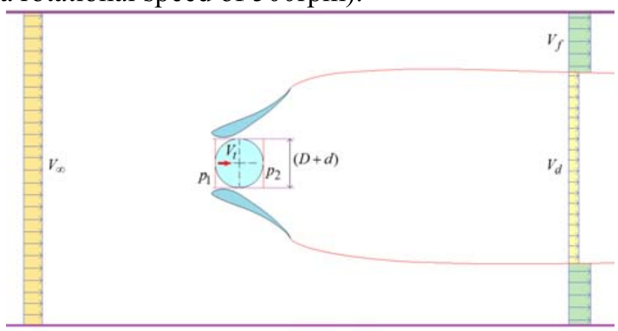

Fig. 3. Flow domain.

The ducted turbine can be placed in an unbounded flow domain, or the flow domain can be bordered by straight side walls as in figure 3.

Across the turbine, there is a pressure drop $\Delta p = (p_1 - p_2)$, which can be derived from the energy law written between upstream, where the flow velocity is V_∞, and downstream, where the flow velocity is V_d : by neglecting the pressure difference $(p_\infty - p_d)$, the pressure drop across the turbine is: $\Delta p \cong \rho \left(V_\infty^2 - V_d^2\right)/2$, where ρ is the water density. The turbine power is $P = (p_1 - p_2) A V_t$, where $A = (D + d)H$ is the turbine vertical cross-section area, d is the blade thickness, and V_t is the average velocity in the minimal flow section (see figure 3).

For ducted cross-flow turbines operating in an unbounded flow domain,

$$V_t = (1 + c_s)(V_\infty + V_d)/2, \tag{1}$$

where c_s is the duct force coefficient, defined as the ratio between duct's drag force and turbine's drag force; for an unshrouded turbine, $c_s = 0$. In a free flow domain, the power coefficient is defined as:

$$c_P = \frac{P}{\rho A V_\infty^3 / 2} = \frac{\left(V_\infty^2 - V_d^2\right) V_t}{V_\infty^3} = \frac{1}{2}(1 + c_s)\left(1 - r^2\right)(1 + r), \tag{2}$$

where $r = V_d / V_\infty$ is the ratio between downstream and upstream flow velocities. The optimum (maximum) power coefficient is obtained when $r = 1/3$, for which $c_{P_{max}} = (16/27)(1 + c_s)$, the result being greater than Betz's limit.

When the flow domain is confined, e.g. bordered by side walls as in figure 3, the blockage coefficient c_b is defined as [21]:

$$c_b = (1 + c_s)(D + d)H / A_\infty \,, \tag{3}$$

where A_∞ is the cross sectional area of the flow domain bordered by walls. The wall blockage effect is inserted in c_P as a correction factor [21], yielding:

$$c_P\big|_{c_b > 0} = (1 + c_s)\frac{r\left(r_f + r\right)^2\left(r_f - r\right)}{2r + r_f - 1}, \tag{4}$$

where $r_f = V_f / V_\infty$ is the ratio between the downstream velocity outside the wake and the upstream flow velocity; here, V_d from $r = V_d / V_\infty$ is the downstream velocity inside the wake.

The power coefficient for an unbounded flow domain (where $c_b \to 0$), can be computed with the following formula [21]:

$$c_P\big|_{c_b = 0} \approx (1 - c_b)^2 c_P\big|_{c_b > 0} \,. \tag{5}$$

The computational effort has been significantly reduced by using an innovative modelling approach adapted to COMSOL Multiphysics as in Georgescu *et al*. [8], an approach that couples a macroscopic non-rotational model of the turbine with a Reynolds Averaged Navier-Stokes (RANS) calculation, using the $k - \varepsilon$ turbulence model in *Rotating Machinery*, *Transient Analysis*. Within that numerical procedure, the effect of the Achard turbine, which turns unshrouded at constant angular velocity ω, is inserted via a *fictitious turbine* inside the channelling device; the *fictitious turbine* acts on the fluid flow like a real turbine, producing a pressure drop Δp as the real one. Thus, the transient flow modelling inside such a ducted fictitious turbine can be performed more than 60 times faster than in the real turbine case. During computations, outside and especially downstream of the fictitious turbine, the flow behaviour is similar to the one of a real turbine; differences with respect to the unshrouded turbine are due to the inter-influence of the duct (channelling device foils) and the turbine.

The first step was to model the transient flow inside an unshrouded Achard turbine. The computational domain was $12D$ long ($5D$ before the turbine) and $8D$ wide. The unstructured mesh consists of 4477 triangular elements, 317 boundary elements, 138 vertex elements, yielding 39471 degrees of freedom. We used two sub-domains: a rotating one (turbine's swept area of diameter ($D + d$)), and a fixed one (outside the former).

The used boundary conditions were: inflow with a specified velocity $V_\infty = 2.3$ m/s, turbulent intensity of 0.05 and turbulence length scale of 0.1 on the left hand side of the domain; zero pressure on the right hand side of the domain; slip symmetry on the upper and lower boundaries; logarithmic wall function with the offset of $h/2$ on the blades; neutral identity pair on the boundaries between the fixed and rotating sub-domains. All computations were performed for 6 full rotations of the turbine; no data was recorded for the first 5 turns of the turbine, and only the 6th complete turn was used to yield results.

We also added to the model 6 *Boundary Integration Coupling Variables* in order to get, at the end of the simulation, the values of the forces on *Ox*- and *Oy*-directions, for each blade, at each time step, as well as a *Sub-domain Integration Coupling Variable*, to get the average velocity U_m in the rotating sub-domain.

From the data recorded for the last complete rotation of the unshrouded Achard turbine, we plotted the values of the *Boundary Integration Coupling Variables* and converted the plots to ASCII files. Those files were processed in Microsoft Excel to obtain the values of the tangential forces F_t acting on each blade at each time step. In fact, at each time step, we integrated pressure values over each blade and we obtained thus the values of the forces acting on each blade with respect to the flow direction (*Ox*) and across the flow direction (*Oy*), denoted F_x and F_y in the sequel, and measured in [N/m], as we used a 2D model; then, for each time step and each blade, those forces were composed to yield the value of the resulting tangential force to the turbine, which is the one responsible for turbine rotation. By adding for each time step the values of the tangential force on the 3 blades and multiplying the result with the turbine radius, we get the value of the thrust on the turbine shaft as a discrete function of time for a full rotation.

It is then simple to average these discrete values of the thrust for a full rotation, to get the average trust acting on the turbine. Finally we multiplied the average thrust with the angular velocity ω of the turbine, and normalized the result by the power of the fluid $0.5\rho D V_\infty^3$ (we neglected here the blade thickness), to get the average power coefficient c_P of the unshrouded Achard turbine. As we used a 2D model, the height of the turbine has no relevance.

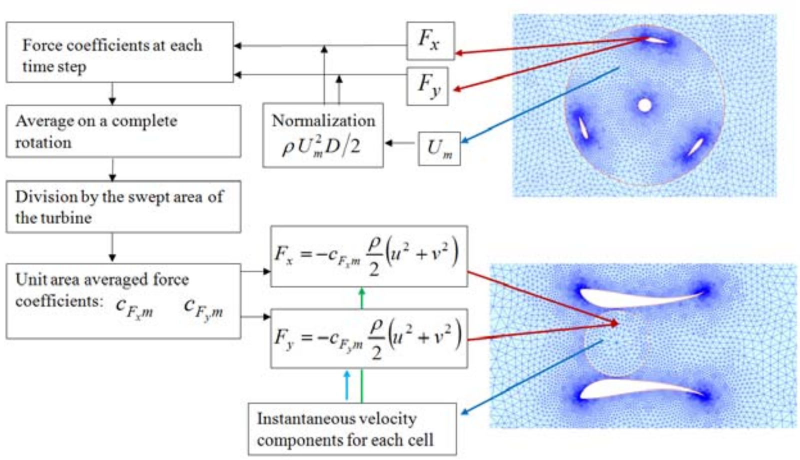

Fig. 4. Scheme of the numerical procedure

The second step was to model the unsteady flow inside a ducted Achard turbine. To reduce the computational effort, the idea was to replace the rotating turbine, by a *fictitious turbine* (a fixed turbine) that would yield on the flow the same effect as the rotating turbine. To do so, in the ducted numerical model (containing the two symmetric S1223-RTL foils of the channelling device), we firstly define a circular sub-domain, corresponding to the turbine swept area of diameter $(D+d)$ and representing the fictitious turbine. Within this fictitious circle, the resulting force corresponding to the rotating turbine is spread as unit volume force (or unit area force in the case of 2D simulations) over the whole non-rotational circular domain. Inside the non-rotating domain of the fictitious turbine, we cannot expect to obtain flow behaviour somehow similar to the one of the rotating turbine – in fact, the fictitious turbine represent an average over a full rotation of the real turbine. The procedure that we used is depicted in figure 4.

From the unshrouded Achard turbine simulations, we computed unit area averaged force coefficients on *Ox*- and *Oy*-directions ($c_{F_x m}$ and $c_{F_y m}$), which were considered as constants in the ducted model. To do so, we considered the forces acting on the unshrouded turbine, F_x and F_y, as known for each time step, and each of the blades. We added the forces on corresponding directions (*Ox* and *Oy*) on the three blades for each time step.

Then, we integrated the velocity field on the moving mesh domain, for each time step, and divided the resulting values of the integrals by the area of the moving mesh domain, in order to get average velocities U_m over the moving mesh domain at each time step. We normalized the values of the forces on both directions by the term $0.5\rho D U_m^2$ and got force coefficients on the turbine for each time step. We averaged those coefficients on a complete rotation, to get average force coefficients on the turbine. Finally we divided the resulting averaged coefficients by the area that represents the fictitious turbine in the ducted model: the area of a circle with diameter $(D+d)$. The resulting values represent unit area averaged force coefficients. For all the following calculations, we considered those coefficients as constants.

Finally, according to the action-reaction principle, we add in the circular non-rotational sub-domain of the fictitious turbine the volume forces computed as: $F_x = -c_{F_x m}\rho\left(u^2+v^2\right)/2$, and $F_y = -c_{F_y m}\rho\left(u^2+v^2\right)/2$, where u and v are the two local cell velocity components on the two directions *Ox* and *Oy* respectively (i.e. resulting from each iteration of the solver, for each cell in the sub-domain).

Thus, we will get outside the fictitious turbine area, a mean flow similar to the one produced by a rotating turbine, averaged over a complete rotation.

According to the 6 incidence angle values, listed in Section 1, we tested 6 ducted Achard turbine configurations. The computational domain extension was not the same, due to the opening of the duct outlet with increasing α values: it varied from 3m long (almost $17D$), with 1.4m before the turbine, and 0.7m wide (meaning $4D$) for $\alpha = 0^\circ$, to 12m long (almost $68D$), with 6m before the turbine and 1.6m wide (almost $9D$) for the other α values. For $\alpha = 30^\circ$, the unstructured mesh consists of 5605 triangular elements, 375 boundary elements, 210 vertex elements, yielding 49325 degrees of freedom. The fictitious turbine was separated from the main flow domain by neutral boundaries, while the rest of the boundary conditions were the same as for the unshrouded turbine model.

For each of the above 6 ducted turbine configurations, based on velocity isolevels of 2.3 m/s, the wake can be delimited in COMSOL *Post processing*, as in figure 5a, in order to extract both downstream average velocities: V_d, inside the wake and V_f, outside the wake. The average velocity V_t is computed using the *Sub-domain Integration* over the turbine domain. Then, the duct force coefficient c_s can be obtained from (1), the blockage coefficient c_b resulted from (3), and the power coefficient c_P can be computed from (4), since the studied configuration is confined. Finally, the power coefficient for an unbounded flow domain $c_P\big|_{c_b=0}$ can be obtained from (5).

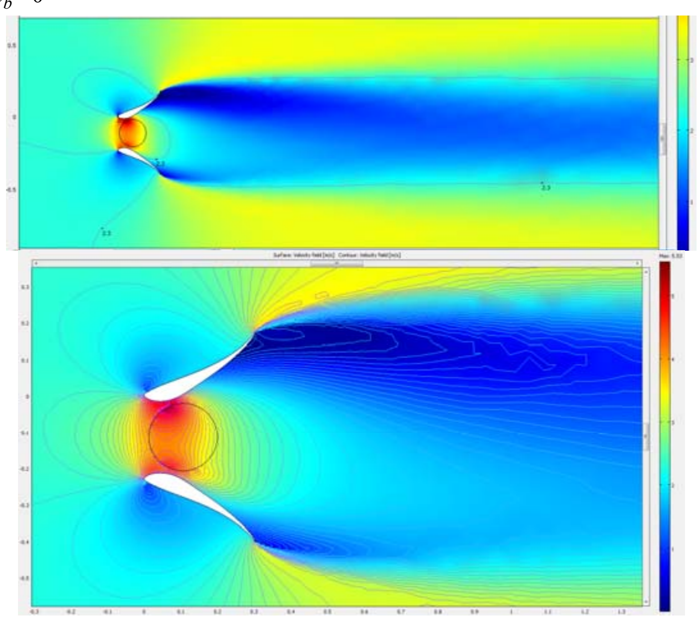

Fig. 5. Velocity field for a ducted Achard turbine with $\alpha = 30^\circ$: velocity isolevels of 2.3m/s (upper frame), and zoomed image with velocity isolines (lower frame).

3. Numerical results

The studied ducted Achard turbine has a diameter $D = 0.175$ m and two S1223-RTL channelling foils of $L = 2D$ chord length. Six numerical models of that ducted turbine were built, in order to quantify the influence of the incidence angle α on the power coefficient c_P, where $\alpha \in \{0^\circ; 14^\circ; 18^\circ; 24^\circ; 30^\circ; 34^\circ\}$. All simulations were performed in a confined flow domain, for a tip speed ratio $\lambda = 2$, with $V_\infty = 2.3$ m/s. In figure 5, we present the velocity field for the ducted Achard turbine configuration with $\alpha = 30^\circ$ and blockage coefficient $c_b = 0.224$.

Table 1

Synthesis of the results for different incidence angle values

α [grd]	Computed in COMSOL Multiphysics				in Fluent [14]	Δc_P [%]
	c_s	c_b	$c_P\big\vert_{c_b>0}$	$c_P\big\vert_{c_b=0}$	$c_P\big\vert_{c_b=0}$	
0	0.351	0.360	0.501	0.205	–	–
14	0.672	0.195	0.697	0.452	0.528	14.5
18	0.763	0.206	0.779	0.491	0.575	14.6
24	0.852	0.216	0.966	0.593	0.621	4.4
30	0.922	0.224	1.072	0.645	0.728	11.4
34	0.939	0.226	1.105	0.661	0.711	7.0

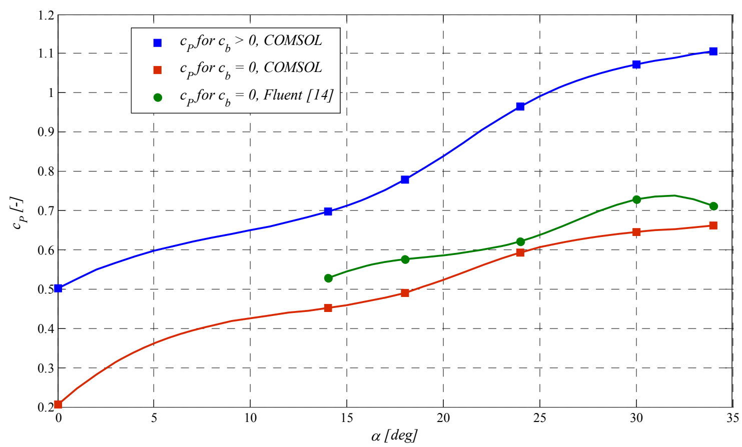

Fig. 6. Power coefficient for the ducted Achard turbine, in confined (4) and free flow domain (5).

Our computed values of the duct force coefficient c_s, blockage coefficient c_b, and power coefficient $c_P\big|_{c_b>0}$ (4) of the ducted Achard turbine were inserted in table 1, together with the resulting power coefficient for a free flow domain

$c_P\big|_{c_b=0}$ (5). We added in table 1 the $c_P\big|_{c_b=0}$ values computed with greater accuracy in Fluent by Menchaca Roa [14], using a $k-\omega$ Shear Stress Transport turbulence model. The relative difference Δc_P [%] in $c_P\big|_{c_b=0}$ values between Fluent and COMSOL results are also inserted in table 1: our computed values are smaller than the ones from [14]. All power coefficients curves $c_P = c_P(\alpha)$ are compared in figure 6, for $c_b > 0$ and for $c_b = 0$.

For the unshrouded Achard turbine, of same size, operating in free flow with $V_\infty = 2.3$ m/s, Menchaca Roa computed in Fluent a power coefficient equal to 0.41 for $\lambda = 2$, that value being the optimum (maximum) one among other investigated cases with λ from 1 to 5 [14, page 43]. For the unshrouded Achard turbine operating in the confined flow domain of the hydrodynamic tunnel of LEGI Grenoble (a 0.7m wide tunnel, meaning $4D$ wide), Menchaca Roa obtained experimentally a power coefficient of 0.387, depicted from [14, fig.3.15, page 61].

4. Conclusions

The 2D numerical modelling of the unsteady flow inside a ducted Achard turbine, operating in a confined flow domain, has been performed using COMSOL Multiphysics, for different incidence angles of the symmetric foils that form the channelling device.

The method used within this study allowed us to determine the power coefficient of the ducted Achard turbine in confined flow domain. The method has proven to save a lot of computational time: e.g. a computation with a rotating turbine would have taken about 2.5 hours on a workstation with 16GB memory and 2 quad-core Intel Xeon 2.66GHz processors, while by using this innovative method, each computation with fictitious turbine took less than 2.5 minutes, meaning running more than 60 times faster. The results concerning the power coefficient of the ducted Achard turbine in free flow domain were compared with similar results obtained by Menchaca Roa [14]: discrepancies of less than 14.6% are acceptable, considering the computational time gain. Our method is useful as first investigation step, to study quickly various geometrical configurations, then the configuration with the best power coefficient can be refined in a second step, using a numerical model of a ducted rotating Achard turbine.

R E F E R E N C E S

[1] *M. S. Güney and K. Kaygusuz*, "Hydrokinetic energy conversion systems: A technology status review", Renewable & Sustainable Energy Reviews, **vol.14**(9), 2010, pp 2996-3004

[2] *B. G. Newman*, "Actuator-disk theory for vertical-axis wind turbines", in Journal of Wind Engineering and Industrial Aerodynamics, **vol. 15**(1-3), 1983, pp. 347-355

[3] *A. S. Bahaj, A. F. Molland, J. R. Chaplin and W. M. J. Batten*, "Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in cavitation tunnel and towing tank", in Renewable Energy, **vol. 32**(3), 2007, pp.407-426
[4] *T. Maître, Ane Mentxaka Roa, C. Pellone and J.-L. Achard*, "Numerical 2D hydrodynamic optimization of channeling devices for cross-flow water turbines", in U.P.B. Sci. Bull., Series D, **vol. 72**(1), 2010, pp. 125-132
[5] *B. Kirke*, "Developments in ducted water current turbines", Sustainable Energy Centre, Univ. South Australia, 2006 http://www.cyberiad.net/library/pdf/bk_tidal_paper25apr06.pdf
[6] *M. S. Güney*, "Evaluation and measures to increase performance coefficient of hydrokinetic turbines", in Renewable & Sustainable Energy Reviews, **vol.15**(8), 2011, pp 3669-3675
[7] *A. R. Malipeddi and D. Chatterjee*, "Influence of duct geometry on the performance of Darrieus hydroturbine", in Renewable Energy, **vol. 43**, 2012, pp 292-300
[8] *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. of the International Conference on Modelling Fluid Flow CMFF'09, September 9-12, Budapest, Hungary, editor J. Vad, **vol. II**, 2009, pp 752-759
[9] *J.-L. Achard and T. Maître*, Hydraulic Turbomachine, FR Patent No. EP1718863(A1), 2006
[10] *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
[11] *Cristina Ploeşteanu*, Étude hydrodynamique d'un type d'hydrolienne à axe vertical pour les courants marins, Institut National Polytechnique de Grenoble, France, 2004
[12] *E. Amet*, Simulation numérique d'une hydrolienne à axe vertical de type Darrieus, PhD Thesis, Institut Polytechnique de Grenoble, France & Technical University of Civil Engineering Bucharest, Romania, 2009
[13] *V. Aumelas*, Modélisation des hydroliennes à axe vertical libres ou carénées: développement d'un moyen expérimental et d'un moyen numérique pour l'étude de la cavitation, PhD Thesis, Université de Grenoble, France, 2011
[14] *Ane Menchaca Roa*, Analyse numérique des hydroliennes à axe vertical munies d'un carénage, PhD Thesis, Université de Grenoble, France, 2011
[15] *T. Jaquier*, Hydroliennes à flux transverse: Développement d'un prototype HARVEST en canal, PhD Thesis, Université de Grenoble, France, 2011
[16] *V. Aumelas, C. Pellone and T. Maître*, "Cavitating behaviour analysis of Darrieus-type cross flow water turbines", in IOP Conference Series: Earth and Environmental Science, **vol. 12**, 012077, 2010 http://iopscience.iop.org/1755-1315/12/1/012077
[17] *Ane Menchaca Roa*, *V. Aumelas, T. Maître and C. Pellone*, "Numerical and experimental analysis of a Darrieus-type cross flow water turbine in bare and shrouded configurations", in IOP Conference Series: Earth and Environmental Science, **vol. 12**, 012113, 2010 http://iopscience.iop.org/1755-1315/12/1/012113
[18] *A.-M. Georgescu, Sanda-Carmen Georgescu, C. I. Cosoiu and N. Alboiu,* "Efficiency of Marine Hydropower Farms Consisting of Multiple Vertical Axis Cross-Flow Turbines", in International Journal of Fluid Machinery and Systems, **vol. 4**, no. 1, 2011, pp 150-160
[19] *A.-M. Georgescu, Sanda-Carmen Georgescu*, *S. Bernad and C. I. Cosoiu*, "COMSOL Multiphysics versus Fluent: 2D numerical simulation of the stationary flow around a blade of the Achard turbine", in Scientific Bulletin of the Politehnica University of Timisoara, Transactions on Mechanics, **vol. 52**, no. 66, Fascicola 3, 2007, pp 13-22
[20] ***, Airfoil Investigation Database, 2012, http://www.worldofkrauss.com/
[21] *M. J. Werle*, Wind turbine wall blockage corrections, http://www.flodesign.org/pdf/BlockagePaper.pdf, 2012.