Device for passive flow control around vertical axis marine turbine

To cite this article: C I Cooiu *et al* 2012 *IOP Conf. Ser.: Earth Environ. Sci.* **15** 062031

Device for passive flow control around vertical axis marine turbine

C I Coşoiu, A M Georgescu, M Degeratu, L Haşegan and D Hlevca

Hydraulics and Environmental Protection Department, Technical University of Civil Engineering of Bucharest, B-dul Lacul Tei, Nr. 124, 020396 Sector 2, Bucharest, Romania

E-mail: costin@hidraulica.utcb.ro

Abstract. The power supplied by a turbine with the rotor placed in a free stream flow may be increased by augmenting the velocity in the rotor area. The energy of the free flow is dispersed and it may be concentrated by placing a profiled structure around the bare turbine in order to concentrate more energy in the rotor zone. At the Aerodynamic and Wind Engineering Laboratory (LAIV) of the Technical University of Civil Engineering of Bucharest (UTCB) it was developed a concentrating housing to be used for hydro or aeolian horizontal axis wind turbines, in order to increase the available energy in the active section of turbine rotor. The shape of the concentrating housing results by superposing several aero/hydro dynamic effects, the most important being the one generated by the passive flow control devices that were included in the housing structure. Those concentrating housings may be also adapted for hydro or aeolian turbines with vertical axis. The present paper details the numerical research effectuated at the LAIV to determine the performances of a vertical axis marine turbine equipped with such a concentrating device, in order to increase the energy quantity extracted from the main flow. The turbine is a Darrieus type one with three vertical straight blades, symmetric with respect to the axis of rotation, generated using a NACA4518 airfoil. The global performances of the turbine equipped with the concentrating housing were compared to the same characteristics of the bare turbine. In order to validate the numerical approach used in this paper, test cases from the literature resulting from experimental and numerical simulations for similar situations, were used.

1. Introduction

In recent years, several studies were performed on the hydrokinetic turbines in order to extract energy transported by the marine or fluvial currents. In 2001, at the LEGI (Geophysical and Industrial Fluid Flows Laboratory), in Grenoble, a new concept of vertical axis, cross-flow, marine current turbine module (the Achard turbine) was proposed [1] and developed within the French HARVEST Project. Related studies on the interinfluence of the Achard type hydraulic turbines were carried out within the THARVEST project [2] at several Romanian universities including UTCB (Technical University of Civil Engineering of Bucharest). Since there are very many similarities with the aeolian turbines, in the mentioned projects, those turbines were generically named hydrolian turbines.

At the end of the last decade, the data gathered within the two projects offered an overview on the performances and limitations of the turbine module [3] and farms [4]. At that point, the research effort

was redirected at augmenting the performances in order to be able to extract a larger amount of energy from the fluid current.

The power supplied by a turbine with the rotor placed in a free stream flow may be increased by augmenting the velocity in the rotor area. The energy of the free flow is dispersed and it may be concentrated by placing a profiled structure around the bare turbine in order to concentrate more energy in the rotor zone.

In the case of the free rotor, the fluid is decelerating as it approaches the turbine. In the case of a turbine-casing assembly, the static pressure inside the concentrator is smaller than the static pressure in the free stream. A suction effect is obtained, leading to an increase in velocity inside the casing. That leads to an increase of the power generated by the turbine placed inside the concentrator, which is significant, since it varies with the third power of the velocity.

Several concepts of concentrating casings for vertical axis turbines were proposed and among these are mentioned Ponta and Dutt [5], Ponta and Jacoviks[6] and Navabi [7]. In 2011, Roa [8] proposed a profiled duct type energy concentrator to be used in conjunction with a vertical axis Darrieus type hydrolian turbine (the Achard A10 model). The numerical predictions were encouraging; the solution insures a 50 % power increase for the single turbine module.

At the same time, at the Aerodynamic and Wind Engineering Laboratory (LAIV) of the UTCB, studies were performed on a concentrating housing to be used for hydro or aeolian horizontal axis wind turbines, in order to increase the available energy quantity in the active section of turbine rotor. Those concentrating housings may be also adapted for hydro or aeolian turbines with vertical axis. This paper presents the numerical approach used to determine the performances of a vertical axis marine turbine equipped with such a concentrating device, in order to increase the energy quantity extracted from the main flow.

The global performances of the ducted turbine were quantified (Moment coefficient C_M and average Power coefficient C_P) and compared to the corresponding values that characterize the bare turbine. In addition, a comparison was made between the global performances of the shrouded turbine and the global performances of the case studied by Roa.

The moment coefficient C_M is given by $C_M = M_Z / 0.5\rho ARU_\infty^2$ where M_Z is the torque computed with respect to the rotation axis of the turbine, ρ is the water density, A is the unit area, R is the radius of the turbine and U_∞ is the velocity in the free stream. The power coefficient C_P is given by $C_P = P / 0.5\rho AU_\infty^3$ where P is the power measured at the turbine shaft. Considering the fact that $P=M\omega$ (where ω is the angular velocity of the turbine) and defining the tip speed ratio as $\lambda = \omega R / U_\infty$, the power coefficient may also written as $C_P=\lambda C_M$.

2. The turbine and the concentrating device

The investigated turbine is a vertical axis cross flow Darrieus type one, just like the Achard A10 model. It has three vertical straight blades, symmetric with respect to the axis of rotation, generated using a NACA4518 airfoil. The chord of the airfoil (c) is equal to 0.15 m. The diameter of the turbine is D= 1 m. The length of the blades (H) is equal to 1 m. The diameter of the turbine shaft is d=0.05 m. The turbine blades are attached to the hub by three straight horizontal wings with a constant cross section identical to a NACA0018 airfoil with the chord equal to c. The angle of attack of the radial support wings is 0°.

The concentrating device that was used, according to Khan et al. [9] classification, is a multiple hydrofoil diffuser. The shape of the concentrating housing resulted by superposing several aero/hydro dynamic effects.

First, it has an interior profile of a convergent-divergent nozzle with a concentrating effect that leads to air acceleration in the zone were the turbine rotor is mounted. Lilley and Rainbird [10] and also Wang et al. [11] proposed a similar concept to be used for horizontal axis wind turbines. The long divergent zone downstream the throat of the convergent-divergent nozzle gradually recovers static pressure against the kinetic term.

Second, the casing design is based on an airfoil with high lift and efficient lift to drag ratio. The upper part of the airfoil is placed at the interior zone of the concentrating device, were the turbine is mounted. The aero/hydrodynamic characteristics of the airfoil insure an intensification of the flow on its upper part due to induced circulation around the airfoil [12]. In the version designed for horizontal axis wind turbines, starting from the airfoil, a ring-wing type wind concentrator was proposed [13]. In the case of vertical axis turbines, the casing consists of two straight vertical wings, aligned with the direction of the flow and symmetrical with respect to the shaft, with the upper part oriented to the rotor zone (see Figure 1).

Third, fluid injecting slots that connect the higher pressure on the exterior of the casing with the lower pressure at the interior of it were provided. The slots ensure water injection in the boundary layer in order to energize it and delay its separation. This leads to reduced pressure losses which insure an increase of the volumetric flow rate that passes through the turbine. A similar approach is found in the papers of Gilbert et al. [14], Igra[15] and Philips et. al. [16], in order to augment wind turbines performances.

Fourth and last, the casing has a high divergent angle downstream the throat which leads to flow separation at the trailing edge of the casing. That provides a lower pressure in the wake, downstream, which implies a volumetric flow rate increase trough the minimal section of the casing. For a small power horizontal axis wind turbine, Abe et al. [17] and Ohya et al. [18] used a flanged diffuser in order to ensure flow separation at the end of the concentrator. The flanged diffuser is strongly decreasing the static pressure downstream the turbine and implicitly generate higher flow through the diffuser.

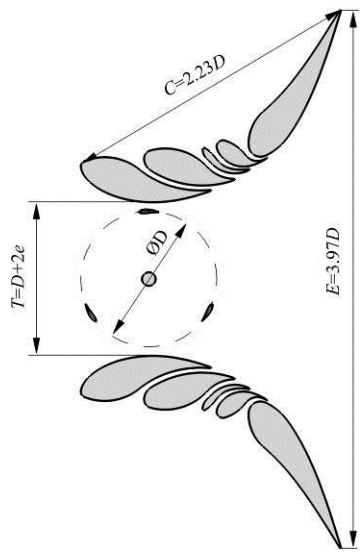

Figure1. The turbine and the concentrating casing

A casing with energy concentrating effect that includes all the favorable effects generated by the above mentioned solutions was proposed. In Figure 1 is presented the casing-vertical axis turbine assembly that was investigated.

For the shape of the concentrator a properly modified NACA4412 airfoil was used. The modification was made in order to have a larger opening angle downstream the throat. The chord C of the airfoil is equal to 2.23 turbine diameters. In the minimum section of the casing, the throat characteristic length is equal to $T=D+2e$, where $e=0.067D$. The distance in the direction of the flow between the leading edge of the airfoil and the rotation axis of the turbine is equal to one turbine radius. The downstream exit opening has a characteristic length $E=3.97D$. To control the boundary layer reattachment, 4 injecting slots were used. Their positions were optimized for a horizontal axis turbine equipped with a ring wing version of the concentrator.

3. Numerical setup

All the numerical simulations were performed using the commercial expert software ANSYS FLUENT. Seven numerical simulations were performed, 3 for the bare turbine and 4 for the shrouded one.

The numerical simulations were conducted considering the hypothesis of identical flow in parallel transversal planes on the turbine rotational axis, so the space dimension was set to be a 2D one. For that, it was considered a horizontal reference plane that intersects the turbine at a level $z=H/4=0.25D$. The computational domain extends horizontally between -5D and 10D, with the origin of the xOy plane placed at the rotation axis of the turbine. For the bare turbine case, the domain in the cross-flow direction covers an interval between -4D and 4D, while for the shrouded turbine case, the domain extends in the cross-flow direction between -16D and 16D.

Table 1. Transient simulation parameters for studied cases

Bare turbine			Shrouded turbine		
ω [1/s]	λ [-]	Δt [s]	ω [1/s]	λ [-]	Δt [s]
2	1	0.08727	2	1	0.08727
4	2	0.04363	4	2	0.04363
6	3	0.02909	6	3	0.02909
-	-	-	8	4	0.02182

Considering the profound unsteady characteristics of the flow around this type of rotating machinery, all the numerical simulations were conducted in a transient time hypothesis. To capture the effects of rotating blades, a sliding mesh model (SMM) was used. A circular fluid zone around the turbine, centered in its axis of rotation was considered to rotate with an angular velocity ω specific to different tip speeds ratios λ. For the bare turbine simulation the rotating fluid zone had a radius of 0.6 m while for the shrouded turbine simulations, the rotating zone has a radius of 0.54 m. The time step Δt was considered, for all performed numerical simulation to be equal with the time necessary for the turbine to rotate with 1°. In Table 1 are presented the transient simulation parameters for the studied cases.

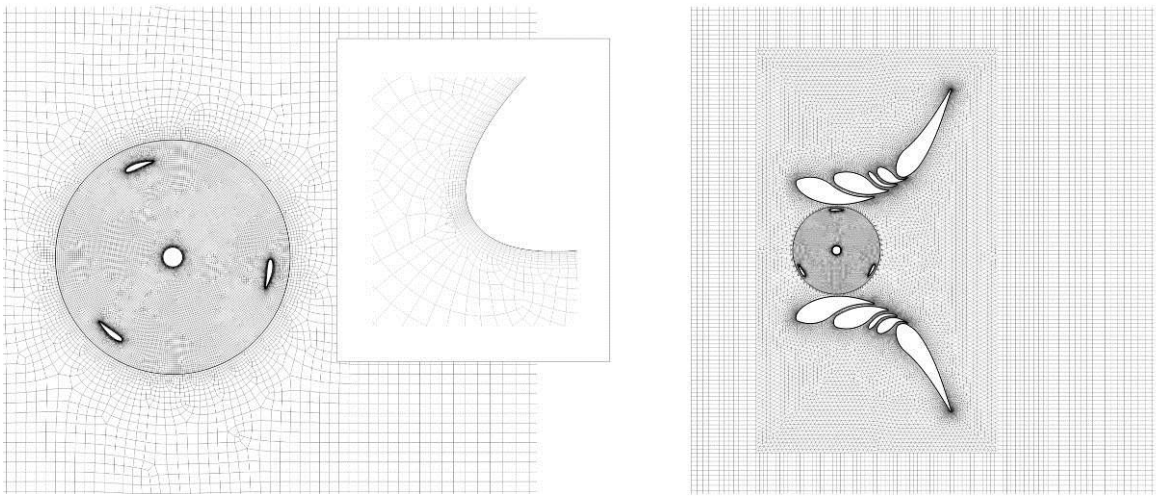

Figure 2. Computational mesh. Bare turbine (left frame) and shrouded turbine (right frame)

The mesh was constructed mainly from structured quad cells. Around the turbine an unstructured grid, made from quad cells was used. Near the casing triangular cells were used. In order to obtain a $y+$ distribution characterized by low values around the walls, a boundary layer discretization near all the solid boundaries was performed. Thus, 8 layers of quad cells near the blades and shaft of the turbine and 15 layers of quad cells near the concentrating casing were used. The total number of cells was equal to 54477 for the bare turbine grid and 118382 for the shrouded turbine model. In Figure 2 the details of the computational meshes are presented.

At the inlet section, placed at the left hand side of the computational domain, a uniform velocity distribution was considered, with a magnitude $U_\infty = 1$ m/s. At the outlet section (right hand side of the domain) the pressure was set to be equal to zero, gage scale. On all solid boundaries, a no-slip condition was considered. The upper and lower frontiers of the computational domain were set to be zero shear slip walls.

In order to solve the turbulent characteristics of the flow the k-ω SST RANS type turbulence model was adopted. For simulations, second order discretization schemes for pressure, the momentum equation, transport equations for the specific turbulence model parameters and time were used. The pressure-velocity coupling has been achieved through a double precision coupled algorithm.

4. Results and discussion

In order to quantify and discuss the performances of the shrouded turbine, the knowledge of the unit operation parameters in the free stream (i.e. without the concentrating device) are needed.

First, numerical simulations on the bare turbine were performed. The values of the forces on the blades as time series were extracted in order to compute moment and power coefficients that characterize the unit. In Figure 3, the variation of the moment coefficient C_M over 360° of rotation for different tip speeds ratios λ is presented. As in previous papers of Georgescu et al. [4] and Roa et al. [19] was found that the optimal operation of the turbine is obtained for a tip speed λ=2; similar patterns for the variation of the moment coefficient were also obtained. The tip speed ratio λ=2 is the only situation where only positive values for the moment coefficient were obtained for each position of the blades during a full rotation.

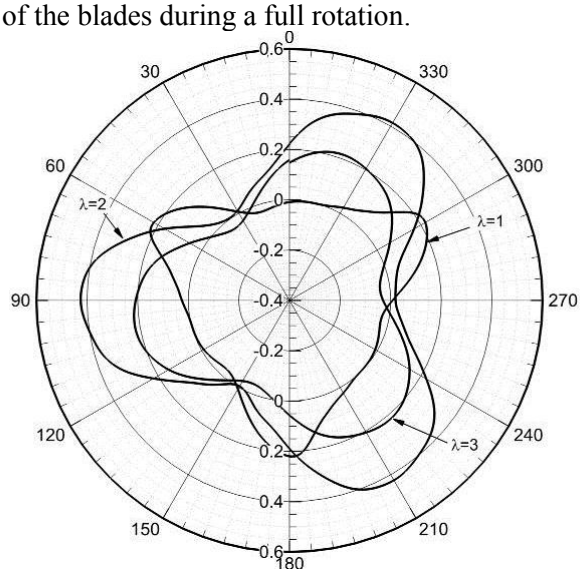

Figure 3. Polar representation of C_M for a full rotation of the bare turbine

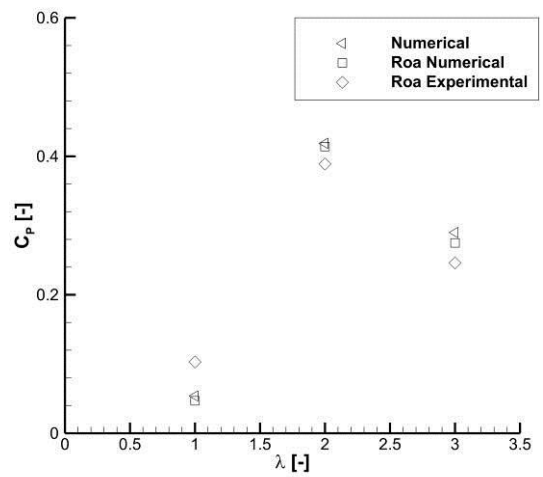

Figure 4. Average power coefficient C_P variation as a function of the tip speed λ for the bare turbine

In Figure 4, power coefficients C_P variation as a function of the tip speed λ is presented. These are average values computed for a full rotation of the turbine. Additional numerical and experimental data from Roa's similar study was used in order to compare the obtained results. A very good agreement between both numerical studies was found. When compared to the experimental data, some differences were observed. The data does not fit very well. The numerical simulations slightly over predict the output power of the turbine. For λ=1 the experiment gives a larger value for the power coefficient. Higher discrepancies were obtained for different tip speeds ratios than the optimal operating point. These difference can be explained by the 2D hypothesis used for space dimension that does not take into account the wake generated by the supporting wings and the tip vortices which decrease global performances of the unit. However, for the present study the differences are considered to be acceptable.

Next, numerical simulations on the shrouded turbine using the proposed concentrator casing, were performed. In Figure 5, moment coefficient C_M variations as a functions of different tip speeds ratios λ are plotted. For λ=1 and λ=2 the moment at the turbine shaft presents strong variation during a full rotation period. The negative value computed at some points suggests that the turbine is working in a braking regime at different moments over a full rotation cycle. For λ=3 and λ=4, a more smooth distribution for the moment coefficients is observed. In addition, only positive values were obtained for the entire full rotation.

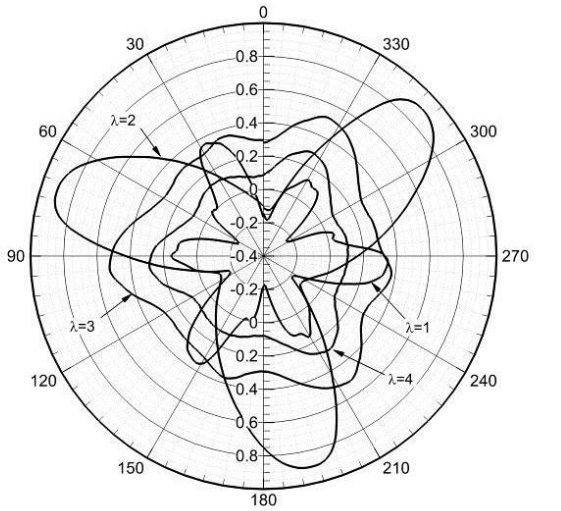

Figure 5. Polar representation of C_M for a full rotation of the shrouded turbine

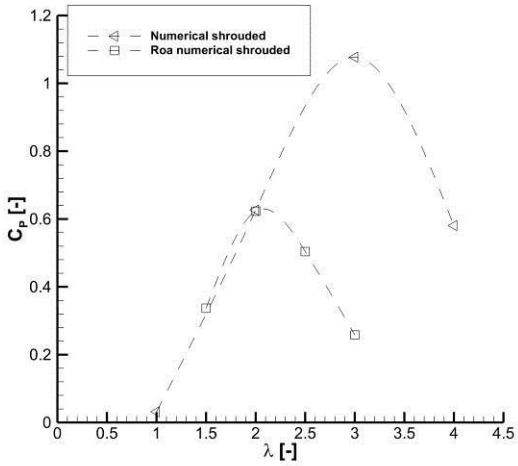

Figure 6. Average power coefficient C_P variation function of tip speed λ for the shrouded turbine

For the case of the shrouded turbine, the optimal operating regime corresponds to a tip speed λ equal to 3. This fact may be observed if the moment coefficient variation over one rotation is examined and also, more obvious, the power coefficient variation as a function of the tip speed, as it is presented in Figure 6.

The maximum value for the power coefficient obtained by Roa's is 0.623 for a tip speed ratio of 2. The maximum value for the power coefficient for the shrouded turbine using the proposed concentrating casing is equal to 1.08, corresponding to an increase with a factor of 1.72. When compared to the bare wind turbine, the power augmentation is even larger, the increase factor being equal to 2.57.

In Figure 7 the vorticity field for the optimal operating regime of the shrouded axial turbine is plotted. The flow field is complicated due to profound unsteadiness of the flow. Larger vortical structures rotating in a counter clockwise direction at the right side of the channelling device (the lower part of the figure) are observed. At the left side of the casing (upper part of the figure) vortical structures rotating clockwise downstream the flow are formed. Behind the turbine shaft, a pattern similar to Karman's vortex street is observed. Those vortical structures are interacting with the channeling casing with a higher intensity at the left side of the flow.

Streamlines plotted in Figure 8 suggests that the flow remains attached at the interior of the casing although the channelling device has a very large opening angle downstream the throat. This is happening mainly due to the slots that are injecting fluid in the boundary layer, re-energizing it and acting as a passive flow control device. Flow separation occurs at the lower part of the casing were the interaction between the channelling devices and vertical structures generated by the rotating turbine is stronger.

The streamlines are also showing a larger amount of fluid that is passing through the turbine. The non dimensional velocity field plotted in Figure 8 suggest a maximal increment of 3.81 that can be obtained by shrouding the turbine (the average value of the increment in the throat being of 2).

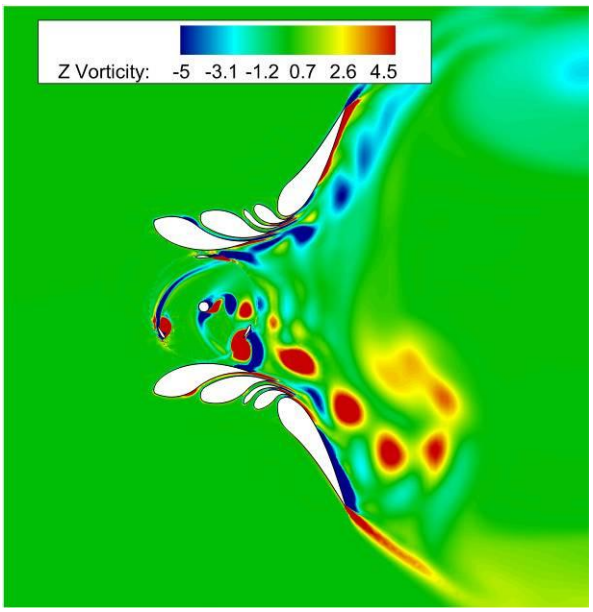

Figure 7. Vorticity field for the shrouded turbine. λ=3

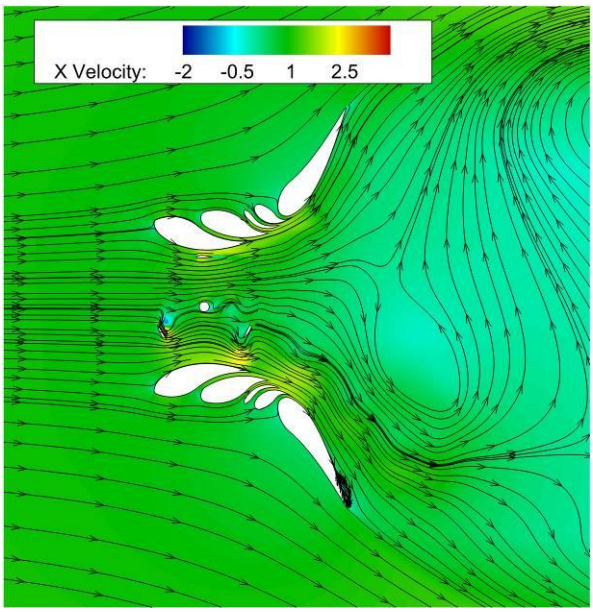

Figure 8. Streamlines superposed over non dimensional longitudinal velocity field for the shrouded turbine. λ=3

5. Conclussions

2D numerical transient simulations for a bare and shrouded vertical axis marine turbine were performed. A multiple hydrofoil diffuser channelling device initially designed for horizontal axis wind turbines was proposed. Its shape resulted by superposing several aero/hydro dynamic effects with favorable consequences on the turbine performances. In order to validate the proposed numerical approach, test cases from the literature resulting from experimental and numerical simulations for similar situations were used.

The numerical simulations predicted an increase in efficiency with a factor of 2.57 for the shrouded turbine with respect to the bare one. The optimal tip speed ratio of the operating unit increased, being equal to 3. A smother distribution for the moment coefficient was also obtained. A larger amount of flow is passing the turbine due to the shrouding which leads to an acceleration of the fluid in the throat of the channelling device.

Acknowledgements

The authors acknowledge the support from the Executive Unit for Financing of the Higher Education, Research, Development and Innovation grants (UEFISCDI PN-II PD-193 105/2010). All numerical simulations have been performed in the Hydraulics and Environmental Protection Department of the Technical University of Civil Engineering, Bucharest.

References

[1] Achard J L and Maître T 2006 *Hydraulic Turbomachine* FR Patent No. EP1718863 (A1)
[2] Georgescu A M, Georgescu S C and Bernad S I 2008 Inter-Influence of the Vertical Axis, Stabilised, Achard Type Hydraulic Turbines – THARVEST, Final Report CEEX Project No. 192/2006, Bucharest, http://www.tharvest.ro
[3] Bernad S I, Georgescu A M, Georgescu S C, Susan-Resiga R and Anton I 2008 Flow investigations in Achard turbine *Proc. of the Romanian Academy* **9**(2) 1-12
[4] Georgescu A M, Georgescu S C, Coşoiu C I and Alboiu N 2011 *International Journal of Fluid Machinery and Systems* **4**(1) 149-159
[5] Ponta F and Dutt G S 2000 *Renewable Energy* **20**(2) 223-241

[6] Ponta F and Jacoviks P 2008 *Renewable Energy* **33**(4) 665-673
[7] Navaby Y 2008 Numerical study of the duct shape effect on the performance of a ducted vertical axis tidal turbine *Ph.D Thesis* (Canada: University of British Columbia)
[8] Roa A M 2011 Analyse numérique des hydroliennes à axe vertical munies d'un carenage *Ph.D Thesis* (French : Université de Grenoble)
[9] Khan M J, Bhuyan G, Iqbal M T and Quaicoe J E 2009 *Applied Energy* **86**(10) 1823-1835
[10] Lilley G M and Rainbird W J 1956 A preliminary report on the design and performance of ducted windmills *Report No. 102, College of Aeronautics*, Cranfield, England
[11] Wang F, Bai L, Fletcher J, Whiteford J and Cullen D 2008 *J. Wind Eng. Ind. Aerodyn* **96** 1–24
[12] Oertel H 2010 *Prandtl-Essentials of Fluid Mechanics* (UK: SpringerLink)
[13] Coşoiu C I, Damian R M, Degeratu M, Georgescu A M and Hlevca D 2011 Numerical study on the efficiency between the ducted and the free stream rotor of a horizontal axis wind turbine *Proc. of the EWEA 2011 Conference (Brussels, Belgium,14-17 March 2011)*
[14] Gilbert B L, Oman R A and Foreman K M 1978 *J. Energy* **2**(6) 368–374
[15] Igra O 1981 *Energy Convers. Manage* **21** 13–48
[16] Phillips D G, Flay R G J and Nash T A 1999 *IPENZ Transactions* **26**(1) 13–19
[17] Abe K, Nishida M, Sakurai A, Ohya Y, Kihara H, Wada E and Sato K 2005 *J. Wind Eng. Ind. Aerodyn* **93** 951–970
[18] Ohya Y, Karasudani T, Sakurai A, Abe K and Inoue M 2008 *J. Wind Eng. Ind. Aerodyn* **96** 524–539
[19] Roa A M, Aumelas V, Maître T and Pellone C Numerical and experimental analasys of a darrieus-type cross flow water turbine in bare and shrouded configurations *IOP Conf. Series: Earth and Environmental Science* **12** 012113