NUMERICAL 2D HYDRODYNAMIC OPTIMIZATION OF CHANNELING DEVICES FOR CROSS-FLOW WATER TURBINES

Thierry MAÎTRE[1], Ane MENTXAKA ROA[2], Christian PELLONE[3], Jean-Luc ACHARD[4]

Lucrarea prezintă o metodologie numerică bidimensională pentru optimizarea unui dispozitiv de carcasare, cu două aripi simetrice profilate, destinat turbinelor cu ax vertical în curent transversal. În prima etapă, curgerea permanentă va fi studiată într-o configurație simplificată, fără rotor, cu o metodă integrală la frontiere şi o analiză a stratului limită. În a doua etapă, rezultatele precedente vor fi validate pentru curgerea nepermanentă în tot domeniul, utilizând ecuațiile Reynolds mediate. Se va arăta că este posibilă reducerea drastică a complexității domeniului în calculele realizate cu instrumentele CFD în procesul de optimizare a geometriei.

This paper aims to present a two-dimensional numerical methodology to optimize a symmetrical two-foiled channeling device for vertical axis cross-flow water turbines. The methodology consists firstly on using a simplified steady approach based on a boundary elements method and a boundary layer analysis for a rotor-less configuration. Secondly, previous results will be validated by full RANS unsteady calculations. It will be shown that it is possible to drastically reduce the amount of geometries to be calculated with CFD tools in the optimization process.

Keywords: cross-flow water turbine, channeling device, CFD, optimization

1. Introduction

Among the renewable energy resources, marine and fluvial currents emerge as one of the most promising options in the near future due to their predictable and endless nature. To harness the kinetic energy contained in these currents, wind turbine concepts have been adapted to operate in an aquatic environment. As a result, two main marine turbine concepts have been developed: the axial-flow type (a transposition in water of commonly used wind turbine); and the cross-flow type (inspired from Darrieus wind turbines). Irrespective of the type, the power output of an isolated turbine is restrained by the Betz limit, which

[1] Assoc. Prof., Laboratory of Geophysical and Industrial Fluid Flows (LEGI), Institut National Polytechnique de Grenoble (INPG), France, thierry.maitre@hmg.inpg.fr
[2] PhD. Student, Laboratory of Geophysical and Industrial Fluid Flows, INPG, France
[3] Senior Researcher, Laboratory of Geophysical and Industrial Fluid Flows, INPG, France
[4] Senior Researcher, Laboratory of Geophysical and Industrial Fluid Flows, INPG, France

corresponds to a power coefficient C_P equal to 16/27 for axial-flow turbines and equal to 16/25 for cross-flow turbines [1]. In reality, whereas axial-flow turbines reach efficiencies up to 45% [2], the reported performance of cross-flow water turbines (CFWT) is only 20-35% [3]. To increase their efficiency, water turbines are equipped with channeling devices that concentrate the fluid flow through the turbine and allow higher energy extraction levels. Methodologies to obtain efficient channeling device are rarely presented and not discussed in detail [1, 4].

In the frame of the HARVEST project, the channeling device benefit is being investigated at the Laboratory of Geophysical and Industrial Fluid Flows (LEGI) of Grenoble. This paper aims to present a 2D numerical methodology to optimize a symmetrical two-foiled channeling device for Darrieus type CFWTs. Considering the high number of parameters defining the channeling device and its associated CFWT on the one hand, and that the numerical modeling of an shrouded turbine is complicated on the other, it appears that the system optimization should require high computational efforts. Though, this study provides a simplified optimization process in two steps that has been numerically validated. The first step consists on using a simplified steady approach based on integral methods (boundary elements methods) for a rotor-less configuration. Different foils are considered at various incidence angles. The channeling device performances are characterized from flow rate coefficients and it is shown that they only depend on the theoretical lift coefficient created by the foils, independently of the geometrical parameters. A boundary layer analysis highlights that theoretical lift coefficients issued from the integral approach can be limited by flow stall on the inner part of the channeling device. This supplementary information is intended to be the definitive criterion to find the best configuration for each specified theoretical lift coefficient. The second step consists on validating the previous one by means of full RANS unsteady 2D calculations of a three-bladed Darrieus turbine equipped with a channeling device.

2. Two-dimensional optimization methodology for channeling devices

This chapter presents the general application of a rough but powerful and very fast numerical technique to optimize CFWT channeling devices: a simplified steady approach based on integral (singularities) methods. Even if this method requires the fluid being inviscid and incompressible, and the flow irrotational, it is a practical calculation providing an approximate solution of a flow around a body or a system of bodies. Calculations are performed on a set of 7 different low drag laminar foils and their corresponding 4-digit NACA foils (foils with the closest similarity in geometrical parameters, as in Table 1). Initially, all the foils are calculated for a fixed incidence of α = 10° (first series). In a second time, for α

values varying unitarily from 0° to 20°, only the EPPLER foils and their NACA foils are calculated (second series).

Table 1

Channeling device foils list

N°	*AIRFOIL*	*CORRESPONDING NACA FOIL*
01	EPPLER-420	NACA 11-4-14
02	EPPLER-421	NACA 09-4-14
03	EPPLER-398	NACA 05-5-14
04	GOE-652	NACA 09-5-17
05	LISSAMAN-7769	NACA 04-3-11
06	S1223-RTL	NACA 08-5-13
07	WORTMANN-FX74-CL5-140-MOD	NACA 10-4-13

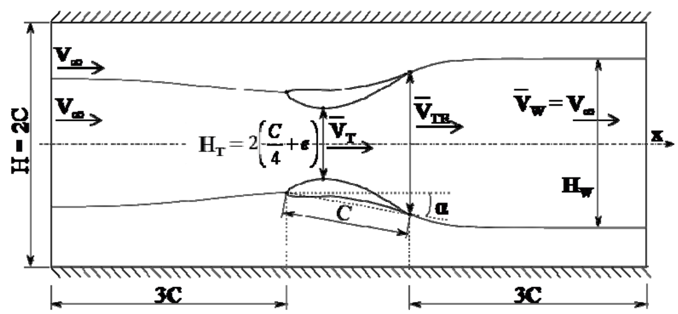


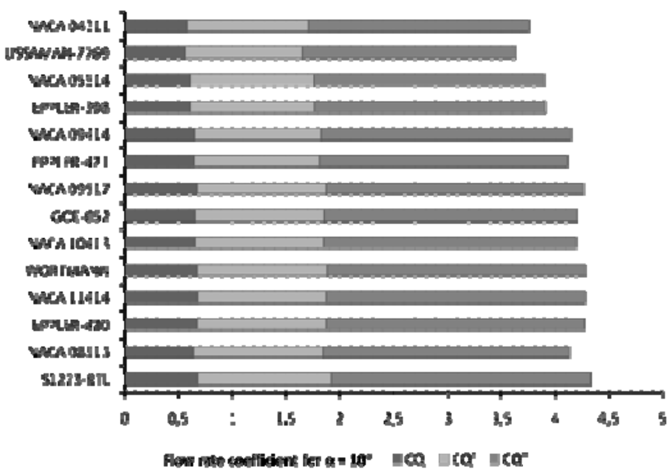
Fig. 1. Schema of the integral method calculations

All calculations have been done using the dimensions of the LEGI hydrodynamic tunnel for further validation of the numerical results with experimental data. The experimental section of the tunnel is 1m long, 0.70m wide and 0.25m high. Flow velocity in the tunnel can be regulated from 0.8m/s to 2.8m/s. For calculations an upstream velocity V_∞ of 1.5m/s has been chosen, which corresponds to rivers average flow velocity. The tunnel is designed to test model CFWTs of 0.175m diameter and 0.175m height. The chord C of the channeling device foils is 0.35m. At this first step, only the channeling device foils are computed numerically. The CFWT is not taken in consideration. The foils are located in order to keep the same vertical distance H_T between the foils (throat spacing), once α is fixed (Fig. 1). H_T takes the rotor dimensions, where $C/4$ is equal to the rotor radius and e designates the gap between the rotor and the foil. The domain length has voluntarily been increased for numerical purposes.

The integral method allows defining different non dimensional coefficients from the calculated pressure and velocity fields, which will determine the device performance. One of the efficiency criteria is the aspiration capacity of the foils because they generate a fluid suction zone upstream the device [3]. This means that there is more fluid passing through the CFWT; extracted energy is therefore bigger. In a first analysis, foils increasing the flow in the rotor are identified. For this purpose, three flow coefficients are introduced: C_Q , C_Q' and C_Q''.

$$C_Q = \frac{Q_{rotor}}{Q_{total}} = \frac{H_W}{H} \qquad C_Q' = \frac{Q_{rotor}}{Q_{size}} = \frac{\overline{V}_{TE}}{V_\infty} \qquad C_Q'' = \frac{Q_{rotor}}{Q_{rotor\infty}} = \frac{\overline{V}_T}{V_\infty} \tag{1}$$

Q_{rotor} is the flow passing through the channeling device, delimited by the stream tube shown in Fig. 1. Q_{total} is the flow passing through the whole domain. Velocity in the far wake section H_W is equal to V_∞ because the flow has already recovered from perturbations. Q_{size} , designated as the obstruction flow, represents the flow at V_∞ passing through a section equal to the distance between the trailing edges of the foils. Finally, $Q_{rotor\infty}$ is the flow at V_∞ passing though a section equal to the vertical spacing in the throat. The lift coefficient (C_{Lift}) is also an interesting parameter that generates the necessary depression for creating the suction zone.



FOIL	CLift at α = 10°
S1223-RTL	4.508128
NACA 08513	4.292124
EPPLER-420	4.460734
NACA 11414	4.575436
WORTMANN	4.429858
NACA 10413	4.320569
GOE-652	4.27535
NACA 09517	4.622252
EPPLER 421	3.944595
NACA 09414	4.126428
EPPLER-398	3.507532
NACA 05514	3.52768
LISSAMAN-7769	2.546058
NACA 04311	2.995535

Fig. 2. Graph and table representing respectively the theoretical flow rate and lift coefficients for the first series at $\alpha = 10°$ calculated with the integral method

Fig. 2 shows the flow rate and lift coefficients obtained for the 1st series at $\alpha = 10°$. The ranking of the most performing foils according to C_{Lift}, C_Q and C_Q'' is: RTL, EPPLER-420 and WORTMANN. Using the C_Q' coefficient, the classification varies slightly: RTL, WORTMANN and EPPLER-420. For the 2nd series of calculations, flow rate coefficients were represented as a function of the theoretical lift coefficient. It was observed that for each flow rate coefficient, all curves were merged. So, results of the 1st series were also represented on the same graph. Fig. 3 illustrates how all results merge in the same curve. It is though concluded that flow rate coefficients only depend on the theoretical lift coefficient provided by the foils, independently of their geometrical parameters. The group of foils, corresponding to high C_{Lift} and high flow rate coefficients, can be selected as the optimal ones at this step. Note that C_Q' is almost a constant curve. So, the rate between Q_{rotor} and Q_{size} is constant, the flow rate traversing the rotor being proportional to the system lateral size (distance between foils trailing edges).

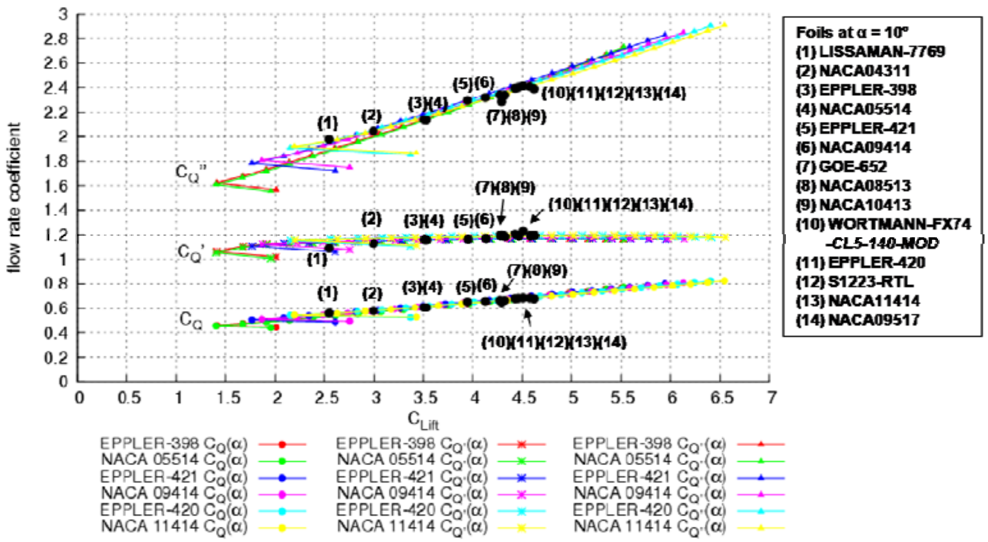


Fig. 3. Universal curve regrouping all numerical results with respect to the theoretical C_{Lift}

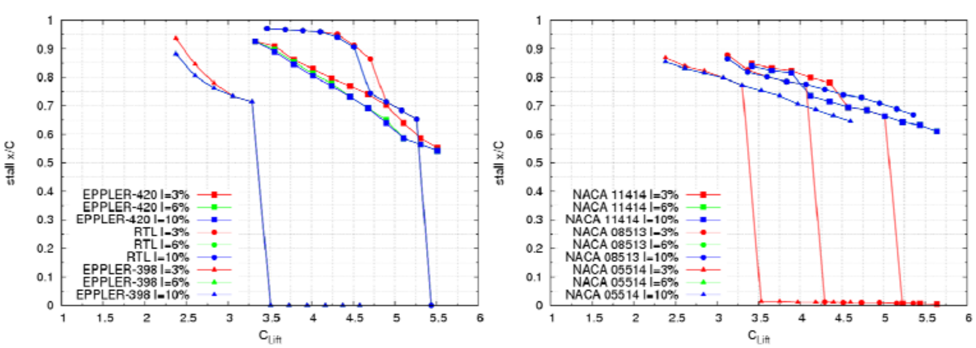


Fig. 4. Stall for the selected foils. Theoretical C_{Lift} corresponds to α values going from 5° to 15°

Well aware that viscosity effects have not been considered in the integral method and that, consequently, possible stalls have not been detected, a boundary layer analysis seems to be obligatory. Thus, a boundary layer code developed at the CREMHyG laboratory was used [5]. Fig. 4 shows the non dimensional boundary detachment position with respect to C_{Lift} for 3 chosen foils and their corresponding NACA. 0 indicates that stall occurs at the leading edge and 1 at the trailing edge. Foils have been chosen from the universal curve: RTL and EPPLER-420 for their good suction capacity and EPPLER-398 for being worse. Stall has been calculated for 3 different turbulence intensities (I): 3%, 6% and 10%. At 6% all foils have reached the saturation level, where stall cannot happen earlier. Note that NACA foils have a smoother behavior at saturation. It is observed that the RTL and EPPLER-420 foils have the best behavior at low I. However, it cannot directly be concluded, at this point, which configuration is the best. At low C_{Lift} (so small α), stall occurs further on the foils, which is theoretically better, but the suction flow is also smaller. Thus it is impossible to say which configuration is the most performing. CFD calculations are required.

3. Validation of the optimization methodology by RANS calculations

Full unsteady RANS calculations have been carried out with the commercial code Fluent to validate the optimization methodology. Calculation domain represents the LEGI hydrodynamic tunnel with the symmetric channeling device, as well as a three NACA 0018 bladed Darrieus model turbine with a 0.032m blade chord. V_∞ has been set to 1.5m/s and the angular velocity ω to operate at the optimal tip speed ratio $\lambda = 4$ [3]. The relationship between the parameters is $\lambda = \omega R/ V_\infty$, with R the turbine radius. The κ-ω-SST turbulence model has been used with I = 3%. Two different power coefficients have been defined for efficiency: $C_{PØ}$, which is the power coefficient with respect to the turbine diameter, and C_{PL}, with respect to the system size (distance between the trailing edges of the device).

The first series of calculations were done with the EPPLER-420 foiled channeling device for α = 6°, 8°, 10°, 12° and 14° corresponding to theoretical C_{Lift} = 3.55, 4.01, 4.46, 4.89 and 5.3. The objective was to verify that, at small C_{Lift} values, efficiency is lower than at high C_{Lift} values, but up to a limit imposed by the boundary layer detachment, which if exceeded, produces a loss in performance. However, calculations seem to show that early stall has no effect. Even at high incidence α for which, according to the boundary layer analysis, stall is supposed occurring closer to the leading edge of the device, efficiency is better and instantaneous torque coefficient values are higher (Figure 5). The maximal α value giving the best performance has not been reached. The 2nd series of calculations were done for 3 foils (EPPLER-420, RTL & EPPLER-398) having the same theoretical C_{Lift} = 4.46, but different stall behavior. Once again, numerical results demonstrate that stall has no effect on the turbine performance when the turbine is taken into account. Figure 6 shows that for the three foils, the instantaneous torque coefficient distribution is almost the same, as well as the efficiency values that are very close. In addition, the same theoretical C_{Lift} value implies having the same flow field independently of the channeling device shape (Fig. 7).

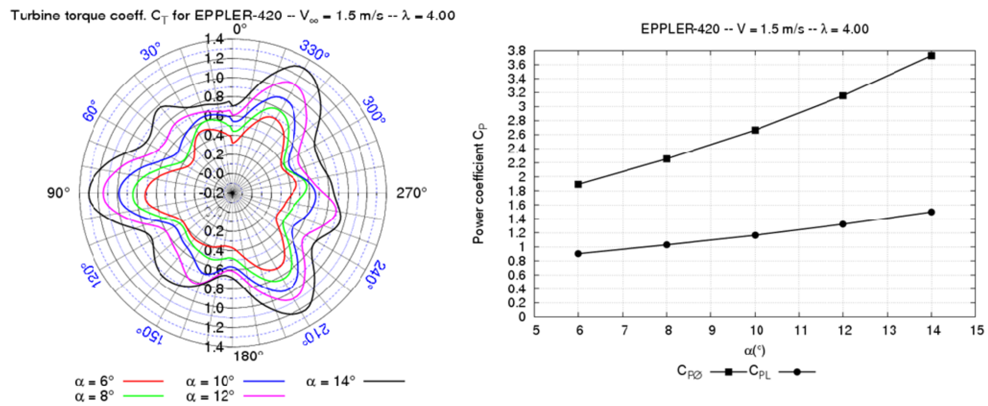
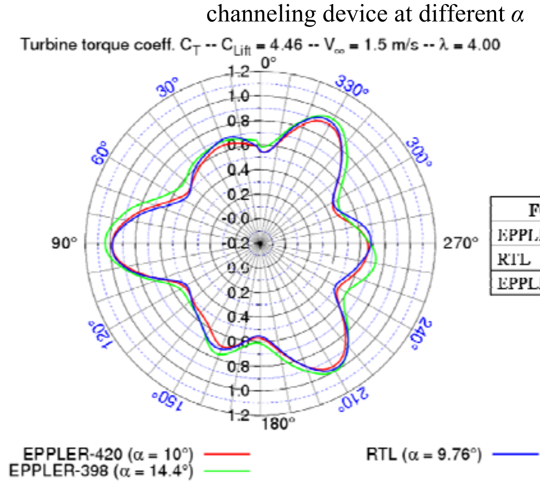


Fig. 5. Turbine torque coefficient distribution (left) and performance (right) for the EPPLER-420 channeling device at different α



FOIL	$C_{PØ}$	C_{PL}
EPPLER-398	2.87	1.23
RTL	2.70	1.22
EPPLER-420	2.67	1.17

Fig. 6. Turbine torque coefficient distribution (left) and performance (right) for the channeling devices for the theoretical C_{Lift} = 4.46

As Ohya and al. stated for an axial flow wind turbine with a flanged diffuser in [6], it is also found in this case for a CFWT placed in the throat of a channeling device that it plays the role of a resistance body and controls the flow separation inside the foils retarding it. This allows having better efficiencies even at high α values. Continuing in the same idea, RANS C_{Lift} should be smaller than the theoretical C_{Lift} due to the flow blockage effect. It could be found a RANS C_{Lift} at high α corresponding to the theoretical C_{Lift} were performance begins to decrease. Further research needs to be done in this axis before concluding definitively on the foils performance limit.

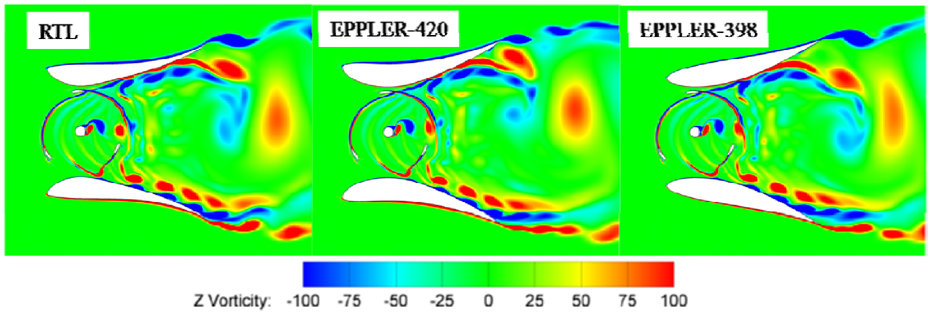


Fig. 7. Vorticity field for the theoretical C_{Lift} = 4.46

6. Conclusion

A 2D numerical optimization methodology for CFWT channeling devices has been developed using inviscid, boundary layer and RANS calculations. Results show that the integral method leads to universal curves where the foils flow aspiration capacity, evaluated by different flow rate coefficients, only depends on the theoretical C_{Lift} created by the foils, independently of their geometrical parameters. The boundary layer analysis applied to the integral method is not correlated to the RANS results. These last ones demonstrate that the channeling device stall behavior is delayed when the rotor is taken into account. Moreover, results prove that, independently of the device geometry, the same flow field is found for the same theoretical C_{Lift}, performances being therefore the same. It can be thought that the CFWT acts as a resistance body modifying the flow behavior. Further research is necessary to better correlate the boundary layer characteristics to the shrouded turbine performance.

REFERENCES

[1]. *B. Kirke*, "Developpments in ducted water current turbines", Sustainable Energy Centre, University of South Australia, www.cyberiad.net/library/pdf/bk_tidal_paper25apr06, 2006

[2]. *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**, 2007, pp.407-426

[3]. *Ane Mentxaka Roa, T. Maître, E. Amet and C. Pellone*, "2D numerical modelling of the power characteristics for cross flow water turbines equipped with a channelling device", in Sci. Bull. "Politehnica" Univ. Timisoara, Trans. Mech., **vol. 53**, no.67, f.3, 2008, pp. 45-56

[4]. *F. Ponta and G. S. Dutt*, "An improved vertical-axis water-current turbine incorporating a channelling device", in Renewable Energy, **vol. 20**, 2000, pp.223-241

[5]. *C. Pellone, T. Maître and L. Briançon-Marjollet*, "Partially cavitating hydrofoils: experimental and numerical analysis", in Journal of Ship Research, **vol. 44**, no. 1, 2000, pp.40-58

[6]. *Y. Ohya, T Karasudani, A. Sakurai, K. Abe and M. Inoue*, "Development of a shrouded wind turbine with a flanged diffuser", in J. Wind Eng. Ind. Aerodyn., **96**, 2008, pp.524-539.