Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 89 (2014) 540 – 547

Procedia Engineering

www.elsevier.com/locate/procedia

16th Conference on Water Distribution System Analysis, WDSA 2014

Experimental Study of Cross-Flow Micro-Turbines for Aqueduct Energy Recovery

V. Sammartano[a], G. Morreale[a], M. Sinagra[a], A. Collura[b], T. Tucciarelli[a,*]

[a]*Università degli Studi di Palermo, Palermo-90128, Italy*
[b]*Istituto Nazionale di Astrofisica, Palermo-90134, Italy*

Abstract

An important component of the management cost of aqueducts is given by the energy costs. Part of these costs can be recovered by transforming some of the many existing energy dissipations in electric energy by means of economic turbines. In this study an experimental work has been carried out: 1) to test the performance of an economic Cross-Flow turbine which maintains high efficiency within a large range of water discharges, and 2) to validate a new approximated formula relating main inlet velocity to inlet pressure. It is proved that the proposed formula, according to some simplifying assumption, exactly links inlet velocity to inlet pressure according to any possible geometry of the Cross-Flow turbine.

Peer-review under responsibility of the Organizing Committee of WDSA 2014

Keywords: Cross-Flow; Banki-Michel; turbines; hydraulic turbine design; hydraulic testing; hydraulic lab.

1. Introduction

Water systems, including extraction of water supplies from natural sources, conveyance, treatment and distribution, end-use, as well as wastewater treatment, require a large amount of energy. The total energy embodied in a unit of delivered water (that is, the amount of energy required to transport, treat, and process a given amount of water) varies with location, source, and use within the country. In most areas, the energy intensity will increase in the future due to limits on water resources and regulatory requirements for water quality and other factors. The rising energy management costs of water supply systems, coupled with local public financial support, are making hydropower

* Corresponding author. Tel.: +39 091 23896518
E-mail address: tullio.tucciarelli@unipa.it

Peer-review under responsibility of the Organizing Committee of WDSA 2014
doi:10.1016/j.proeng.2014.11.476

projects, transforming some of the many existing head dissipations along conduits in electric power, financially attractive for water managers.

Small turbines can be installed potentially in any location where pressure must be reduced in a conveyance system, such as at the head-works of water treatment plants, wastewater treatment plant outfalls or at any pressure-reducing station. Small hydropower projects do not need to be located near a river or a dam and several communities have recently begun to produce electric energy at all existing water facilities [1, 2, 3]. On the other hand, the practical implementation of many plants is still uneconomical due to the need of conventional expensive equipment such as hydraulic turbines, electrical equipment and controllers that had been designed for large hydro power plants and are not suitable for small ones [4].

Cross-Flow (CF) turbines are machines with a simple geometry, design and construction, which display a very good efficiency for very different load conditions. Previous studies addressed the optimal configuration of the CF turbine by using both numerical and experimental methods [5, 6, 7, 8]. Usually, CF turbines are regarded as impulse machines, so that the water flow entering into the impeller behaves like a free water jet leaving the nozzle (such as in Pelton turbines). Using this approach the velocity at the inlet of the impeller should be estimated very close to the Torricelli's formula, as it happens for Pelton turbines [9, 10 ,11], because relative pressure is assumed zero at the impeller inlet. Recent studies [6,7,8] have shown that the pressure at the impeller inlet is far from zero. In the numerical investigation of [8] the CFD analysis showed water pressure at the inlet much larger than zero, leading to a ratio between the Torricelli velocity and the simulated one much lower than 1 (0.75 - 0.85).

In this study a simple relationship between the water head at the inlet, the corresponding mean velocity and the velocity of the reference system at the impeller inlet is proposed. The new formula, as well as the efficiency curve of the turbine designed according to the approach proposed by [7] is experimentally tested using a laboratory plant, specifically designed and constructed in the hydraulic lab of the DICAM Department of the University of Palermo.

2. Design procedure and a new velocity formula

The theoretical approach, described in [7], shows that the maximum efficiency is obtained when the relative tangential velocity is about twice the velocity of the reference system, such that:

$$V \cos\alpha = 2 \cdot \omega \cdot R_1 \tag{1}$$

where V is the velocity norm at the impeller inlet, α is the attack angle, ω is the rotational velocity of the impeller and R_1 is the outer radius of the impeller (see the geometrical scheme of the CF turbine in Fig .1). The turbine prototype was designed assuming α = 22°, which provides a good equilibrium between hydraulic efficiency and structural strains, as also suggested in the literature [5,7]. The following Torricelli's relationship between the impeller inlet velocity and the net head H immediately before the turbine was assumed:

$$V = C_T \sqrt{2gH} \tag{2}$$

where the velocity coefficient C_T would be close to 1 if a zero pressure condition were verified at the impeller inlet and no energy losses occurred inside the nozzle. To design the prototype of the CF turbine the authors selected in a first step few parameter values on the basis of Eqs. (1) and (2), as well as of the continuity equation and the input Q and H data. In a second step other design parameters have been selected with the help of CFD analysis (number of blades and diameter ratio). The radius R_1 was estimated by using Eqs. (1) and (2), given an initial value C_T equal to 0.98 (as suggested in [5-7]), to be iteratively corrected using CFD simulations.

A more precise relationship between H and V can be predicted only if an estimation is made of the pressure p_{in} at the impeller inlet. In this case, assuming steady-state condition and neglecting energy losses inside the nozzle, the following relationship is given by the Bernoulli equation:

$$H = \frac{p_{in}}{\gamma} + \frac{V^2}{2g} \tag{3}$$

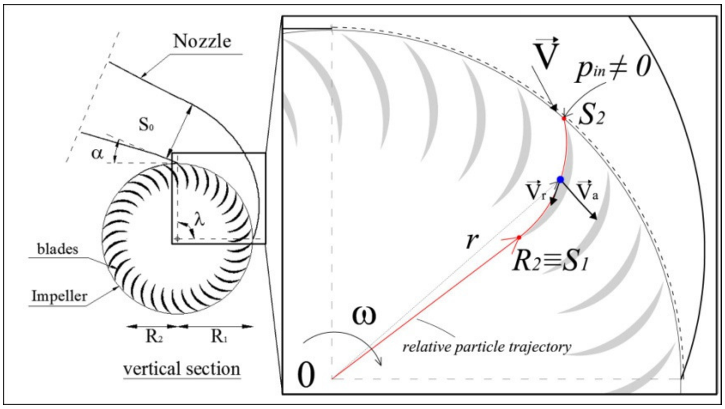

Fig. 1. Geometrical scheme of the CF turbine and of particle moving into the rotating reference system.

where γ is the water specific weight. A rough estimation of the p_{in} pressure can be found by looking at the particle trajectory located next to the inner blade surface and passing through the center of the impeller, assuming radial symmetry. Also, assume a constant norm of the relative velocity of the particle moving inside the rotating reference system and neglect energy losses. This particle, moving along the blade surface, will be subject to an inertial force per unit volume equal to:

$$\mathbf{f} = -\rho\left(\frac{d\mathbf{V}_r}{dt} + \frac{d\mathbf{V}_a}{dt} - 2\boldsymbol{\omega} \wedge \mathbf{V}_r\right) \tag{4}$$

where $\boldsymbol{V_r}$ is the relative velocity of the particle, $\boldsymbol{V_a}$ is the velocity of the reference system at the particle location and $\boldsymbol{\omega}$ is a vector normal to the trajectory plane and with norm equal to the rotational velocity ($\wedge$ is the product operator). The first and the third term in the r.h.s. of Eq. (4) are normal to the blade surface and are balanced by the solid wall reaction. The second term has the radial component of its norm equal to the centrifugal force per unit volume, given by:

$$\rho\left|\frac{d\mathbf{V}_a}{dt}\right| = -\rho\omega^2 r \tag{5}$$

where r is the distance of the particle from the impeller axis. The component of the second term normal to the blade surface, computed according to Eq. (4), is also balanced by the solid wall reaction, but the component tangent to the same surface has to be balanced by a pressure gradient component along the relative trajectory direction. Assuming an exit angle equal to 90°, such that the fluid particle leaving the blade is directed toward the impeller axis, the same force acts on the particles located from the exit point of the blade up to the axis. Assuming zero pressure in the axis, the pressure at the inlet point of this trajectory can be computed as (see Fig. 1):

$$\frac{p_{in}}{\rho} = \int_0^{R_2} \omega^2 r dr + \int_{s_1}^{s_2} \omega^2 r \cos\alpha \, ds = \frac{\omega^2 R_1^2}{2} \tag{6}$$

Merging Eq. (6) in Eqs. (2) and (3) we get:

$$V = C_V \sqrt{2g \cdot \left(H - \frac{\omega^2 R_1^2}{2g} \right)} \tag{7}$$

where C_V is the velocity coefficient, close to 1 only if all the previously mentioned hypothesis (steady-state conditions, radial symmetry, etc...) were exactly satisfied.

3. Experimental Facility and Test descriptions

Experiments have been carried out in the laboratory of the Department of Civil, Environmental, Aerospace, Materials Engineering (DICAM) of the University of Palermo (Italy). The experimental facility is made of a water pumping system and a test stand with the CF turbine prototype coupled with a synchronous generator. A small 5 KW turbine prototype, designed according to the previously mentioned criteria and constructed inside the INAF laboratory of Palermo has been installed in a looped hydraulic circuit. In Fig. 2 a geometrical sketch of the experimental facility is shown.

The pumping system provides the inlet pressure and the flow rate for the CF turbine. The system operates in a closed loop mode and is composed by: an open tank reservoir, a suction pipe, a centrifugal pump, a discharge pipe, a by-pass pipe and two manual valves which allow to regulate both the flow rate and the pressure for the turbine inlet. The flow rate in the discharge pipe is measured by a clamp-on flow meter located in a section shortly upstream the inlet of the CF turbine. The device is a transit time ultrasonic flow meter designed to work with clamp-on transducers.

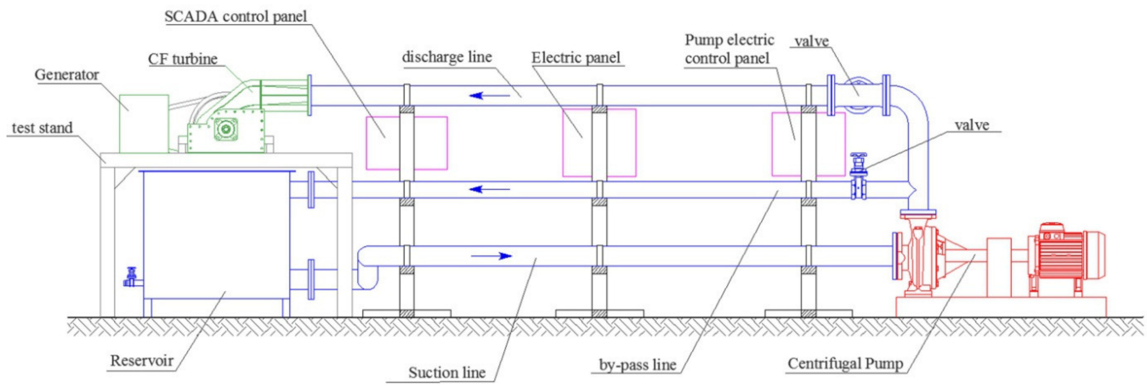

Fig. 2. A geometrical sketch of the experimental facility.

The test stand is made of the CF turbine prototype coupled with a generator, a torque-meter and a digital manometer. The turbine prototype used in the experiments was built at the laboratory of National Institute for Astrophysics (I.N.A.F.) of Palermo (Italy). The turbine prototype has the following main components: the nozzle, the impeller and the casing. The impeller has no internal shaft, and is made by two stainless steel circular plates linked by 35 blades optimized in order to maximize the transfer of energy, according to the previously mentioned procedure. A picture of the CF turbine prototype is reported in Fig. 3.

The axis of the turbine moves a synchronous generator and the produced electric power is dissipated trough resistances. The generator has an horizontal axis and 4 poles rotor, such that with 50 Hz frequency it has a rotational speed of 1500 R.P.M. The generator is fed by direct current (DC) by means of a regulated power supply that converts the alternating current (AC) into a constant DC; this power supply system allows to change the torque at the shaft end and so its rotational velocity. A torque-meter, which allows to measure the torque (up to 200 Nm) transmitted from the turbine shaft to the generator shaft, is installed between the turbine and the synchronous generator. The torque-meter also allows to measure the rotational velocity of the turbine shaft (up to 10000 R.P.M.).

Fig. 3. Two different views of the Cross-Flow turbine prototype.

The experimental stand allows to 1) measure both pressure and pipe velocity immediately above the turbine and provide experimental evidence of the previous Eq. (7), 2) measure the hydraulic efficiency of the turbine and compare its value with the result of the CFD simulations. The acquisition of the experimental data is controlled by a programmable logic controller (PLC) embedded with two I/O compact modules (for 8 analog signals and for 16 digital signals) and a two channel encoder. The software allows to have a real-time access to all the sensors. A Labview code has been developed to process and store the time series of the water discharge flowing into the turbine, the pressure at the inlet, the torque at the impeller shaft and the rotational speed of the impeller.

The tests have been carried out manly for the validation of the efficiency curve and of the Eq. (7) between inlet head and C_V coefficient, as computed with the CFD simulations. Other tests have been carried out in order to study the effect of an internal shaft on the turbine efficiency. To this end, using the same impeller with 35 blades, two different configurations have been compared: 1) without the rotating shaft; 2) with a 30 mm diameter shaft. The experimental runs have been performed keeping a constant value of the pressure p_{up} at the nozzle inlet section (in the range 0.3 - 0.7 bar) and exploring a wide range of rotating velocity, from 300 to 850 R.P.M.. For each test the time series of the water discharge Q flowing into the turbine, the pressure at the nozzle inlet p_{up}, the torque T of the impeller shaft and the rotational speed ω of the impeller have been recorded. The collected data have been used to estimate the turbine efficiency η as ratio between the hydraulic power P_h and the mechanical power P_m at the impeller shaft.

The time series of the aforementioned variables have been filtered in order to reduce the noise of the signal and to remove the outliers values without losing the signal shape. First the data have been de-noised with a cubic smoothing *spline* by using a Fortran routine implemented in a Matlab script; a frequency analysis of the smoothed signal, by means of the Fast Fourier Transform (FFT) algorithm [12] was then used for result validation. The outliers observations were removed by using the whitening test of the residuals $r(t)$ between the original signal $f(t)$ and the smoothed one $f_s(t)$. Thus, the autocorrelation of the residuals $\rho(r,k)$ was estimated in order to identify the data falling into a confidence interval of the 95% coverage [13]:

$$\rho(r,k) = \frac{1.96}{\sqrt{N_k}}, \quad \forall k > 1 \tag{8}$$

where k is the adopted lag-time. After the statistical analysis was carried out on the collected data series, the time averaged values of the velocity coefficient C_V and of the turbine efficiency have been finally estimated by means of Eqs. (7).

4. Experimental and Numerical results

The power P_m measured at the shaft of the turbine is function of the machine geometry and material, as well as of both the discharge Q and the head H at the inlet of the nozzle. On the other hand, the geometric, kinematic and dynamic similarity occurring among experiments with the same velocity ratio V_t/U implies that dimensionless efficiency and velocity coefficient C_V should be function only of V_t/U [14, 15]. For each impeller configuration (with or without internal impeller shaft) the tests have been carried out with a velocity ratio in the range $V_t/U = 1.2 – 2.2$ and an inlet nozzle pressure p_{up} in the range of 0.3 - 0.7 bar. The experimental results have been compared with numerical simulations carried out by means of the commercial solver CFX.

In the case of impeller without the internal shaft the measured efficiencies were almost the same for a constant value of the relative velocity, in agreement with the similarity laws, and the efficiency was always greater than the one calculated by means of CFD analysis. In Fig. 4 the efficiency of the turbine, estimated in each experimental run and in each numerical simulation, is plotted versus the velocity ratio V_t/U. The plot shows that the turbine has a maximum efficiency, η = 82.1 %, for a velocity ratio close to 2 (V_t/U = 1.8), in agreement with the aforementioned design procedure and with Eq. (1). In Fig. 5 the velocity coefficient C_V evaluated by means of Eq. (7), from recorded experimental data as well as numerical simulations, is plotted against the velocity ratio. The regression lines of both the experimental and simulated data sets provide an excellent validation of Eq. (7), with a C_V coefficient bounded by the 0.95 and 1.0 values and only weakly affected by the actual velocity ratio. The linear regression of the estimated velocity coefficient is represented by an almost horizontal line, even if some spreading occurs; the coefficients of regression are $R^2 = 0.083$ for the experimental values and $R^2 = 0.065$ for the numerical simulations.

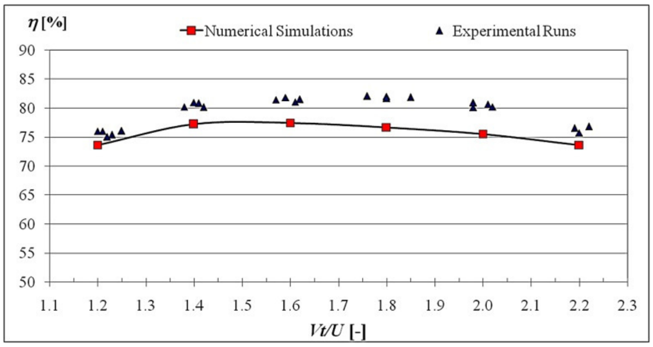

Fig. 4. Plot of the efficiency versus the velocity ratio V_t/U. Comparison between the experimental and the numerical results.

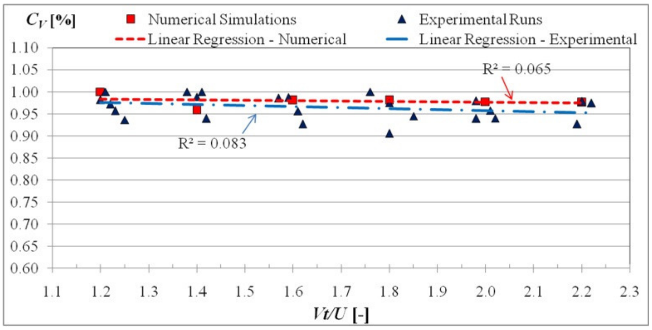

Fig. 5. Plot of the coefficient C_V versus the velocity ratio V_t/U. Comparison between the experimental and the numerical results.

A second set of experimental runs were performed to compare the efficiencies of the turbine with and without the rotating shaft. The experimental runs were carried out with a constant inlet nozzle pressure p_{up} = 0.4 bar and for the same range of velocity ratio adopted for the first set of tests. Thus, the results of these tests have been compared with the results of the no shaft configuration for the same pressure condition (p_{up} = 0.4 bar). The plot of the efficiencies versus the velocity ratio of these tests is reported in Fig. 6.

The plot shows that the trend of the efficiency curves is quite similar, but the performances are lower for the impeller with the internal shaft, in agreement with the numerical results shown by [7]. The coefficient velocity estimated for both configurations, with and without shaft, and for a constant inlet nozzle pressure p_{up} = 0.4 bar, are compared in Fig. 7. The plot shows that the C_V coefficient is almost constant for different values of the velocity ratio and attains values close to one.

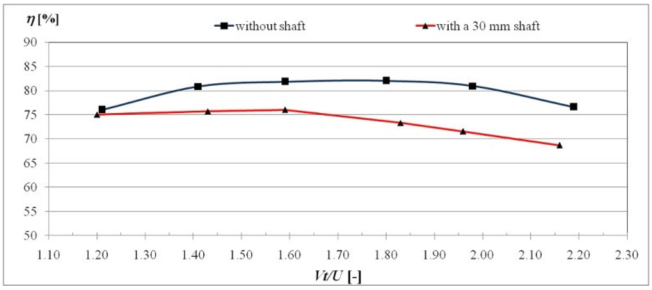

Fig. 6. Comparison of the efficiencies estimated experimentally between the prototype without and with the internal shaft.

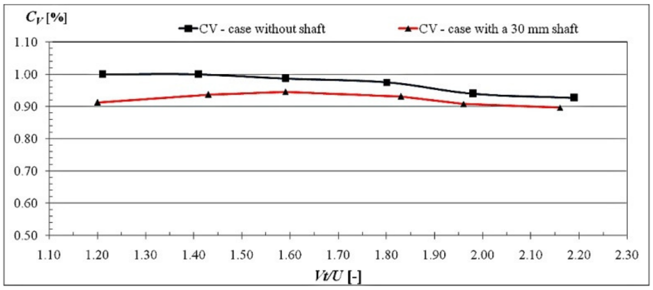

Fig. 7. Comparison of the estimated C_V between the prototype without and with the internal impeller shaft.

5. Conclusions

The previously presented experimental analysis allowed to validate the design procedure of a Cross-Flow turbine, as already developed in [7]. Two different set of tests have been carried out: the first set allowed to verify the efficiency of the designed turbine and to validate the proposed Eq. (7) between the impeller's inlet velocity V and head H; a second set of tests allowed to better understand the effect of the internal shaft on the turbine efficiency. The results can be summarized in the following ones:

- Experimental tests provide a good validation of the results obtained by means of CFD analysis: the trend of the efficiency curves is the same, even if the experimental efficiency values are always a bit greater than the ones obtained by numerical simulations.
- The velocity coefficient CV, estimated by means of Eq. (7), is very close to 1 for all the experimental runs and it is almost independent from the velocity ratio. This means that by evaluating the inlet velocity using Eq. (7) it is possible to perform a good design of the turbine geometry, without iterating the same design according to the actual relationship occurring between inlet velocity V and head H.

- Both experimental tests and CFD simulations show that internal shaft provides a consistent reduction of the machine efficiency, due to the increased internal energy losses. Even if structural analysis is not within the scopes of this research, we observe that turbine structure has not shown any evidence of deterioration after several days of laboratory testing.

References

[1] C. Crampes, M. Moreaux, Water resource and power generation, International Journal of Industrial Organization, 19(6) (2001) 975-997.
[2] A. Demirbas, Focus on the world: Status and future of hydropower. Energy Sources Part B-Economics Planning and Policy, 2(3) (2007) 237-242.
[3] A.A. Williams, R. Simpson, Pico hydro – Reducing technical risks for rural electrification, Renewable Energy, 34 (2009) 1986-1991.
[4] X. Yang, Y. Song, G. Wang G, W. Wang, A Comprehensive Review on the Development of Sustainable Energy Strategy and Implementation in China, Ieee Transactions on Sustainable Energy, 1 (2010) 57-65.
[5] N.M. Aziz, V.R. Desai, A laboratory study to improve the efficiency of cross-flow turbines, Engineering Report, 1993.
[6] J. De Andrade, C. Curiel, F. Kenyery, O. Aguilln, A. Vásquez, M. Asuaje, Numerical investigation of the internal flow in a Banki turbine, International Journal of Rotating Machinery, 2011.
[7] V. Sammartano, C. Arico, A. Carravetta, O. Fecarottam, T. Tucciarelli, Banki-Michell Optimal Design by Computational Fluid Dynamics Testing and Hydrodynamic Analysis. Energies, 6(5) (2013) 2362-2385.
[8] M. Sinagra, V. Sammartano, C. Aricò, A. Collura, T. Tucciarelli, Cross-Flow turbine design for variable operating conditions, In 12th International Conference on Computing and Control for the Water Industry, CCWI2013, Procedia Engineering, 70 (2014) 1539-1548.
[9] J. A. Chattha, M.S. Khan, S.T. Wasif, O.A. Ghani, M.O. Zia, Z. Hamid, Design of a cross flow turbine for a micro-hydro power application, in, Chicago, IL, 2010, pp. 637-644.
[10] J. Zanette, D. Imbault, A. Tourabi, A design methodology for cross flow water turbines, Renewable Energy, 35 (2010) 997-1009.
[11] O. Zia, O.A. Ghani, S.T. Wasif, Z. Hamid, Design, fabrication and installation of a micro-hydro power plant, Report 2010.
[12] M. Frigo, S.G. Johnson, FFTW: an adaptive software architecture for the FFT, in: Acoustics, Speech and Signal Processing, 1998. Proceedings of the 1998 IEEE International Conference on (Volume:3), Seattle, WA, 1998, pp. 1381 - 1384 vol.1383.
[13] P.J. Brockwell, R.A. Davis, Time Series: Theory and Methods., Springer-Verlag, New-York, 1991.
[14] H. Ramos, Guidelines for Design of Small Hydropower Plants., in, CEHIDRO, WREAN and DED, Belfast, North Ireland, 2000.
[15] A.K. Seppo, Principles of Turbomachinery, 2012.