International Journal of Engineering

Journal Homepage: www.ije.ir

Numerical and Experimental Investigations for Design of a High Performance Micro-hydro-kinetic Turbine

A. Zahedi Nejad[a], M. Rad[b], M. Khayat*[a]

[a] Department of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran
[b] Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran

PAPER INFO

Paper history:
Received 05 June 2016
Received in revised form 07 February 2017
Accepted 10 February 2017

Keywords:
Duct
Dynamic Simulation
Fabrication
Hydro-kinetic Turbine
Propeller Test

ABSTRACT

Design and manufacturing of a high performance micro-hydro-kinetic turbine is discussed in the present paper. The main goal is manufacturing an equipped experimental model of hydro-kinetic turbine with highest energy absorption from water current. A multi-shape ducted turbine comprised of a multi-part diffuser was manufactured that can be converted to many experimental models for studying various diffuser enhancing effects. Turbine's rotor included a three-blade axial propeller and a mixed six-blade propeller with high power coefficient. Simple experiments on propeller were performed for flow visualization, torque measurement and illustrating dynamic balance at high speed rotation in air and water. Important data for design and manufacturing of duct and rotor components that led to safe structure and balanced the rotor at high speed rotations were discussed. For dynamic simulation of turbine, a user-defined function was developed for ANSYS-FLUENT software that collects integration data and solves rotor's dynamic equation in one-degree of freedom motion. Many stable dynamic simulation methods for coupling with transient one-dimensional flow around one-degree of freedom propellers were proposed and the numerical results were validated against full CFD data.

***doi**: 10.5829/idosi.ije.2017.30.05b.19*

NOMENCLATURE

A_e	Exit area of duct (m^2)	Re_D	Reynolds number based on average duct diameter
A_i	Inlet area of duct (m^2)	T_0	Propeller torque
A_{th}	Throat area of duct (m^2)	u_0	Area-averaged flow velocity (m/s)
C_D	Drag force coefficient	u_{th}	Area average throat velocity (m/s)
C_p	Power coefficient	W	Net electrical input power (Watts)
D	Average duct diameter (m)		
E	Input electrical voltage (V)	**Greek Symbols**	
I	Input electrical current (A)	ω (R.P.M)	Propeller angular speed
P	Absorbed power (W)	$\lambda = R_0\,\omega / u_0$	Tip-speed ratio
P_{mech}	Mechanical output power of dynamo-motor		
R_0	Propeller radius (m)		

1. INTRODUCTION

During past decades, manufacturing of industrial hydro-kinetic turbines has experienced great growth. Energy absorption from river flows and tidal currents has been commercialized in many countries. Experimental works of most researchers have illustrated the effect of propeller and duct on the energy absorption of hydro-kinetic turbines. In 1993, Quamrul et al. investigated geometric characteristic for blades of horizontal axis wind turbines at different wind conditions [1]. In 2004, Setuguchi et al. designed and manufactured a diffuser-enhanced duct with two passages [2]. They introduced duct shape as a key factor for increasing the efficiency

*Corresponding Author's Email: *mkhayat@srbiau.ac.ir* (M. Khayat)

Please cite this article as: A. Zahedi Nejad, M. Rad, M. Khayat, Numerical and Experimental Investigations for Design of a High Performance Micro-hydro-kinetic Turbine, International Journal of Engineering (IJE), TRANSACTIONS B: Applications Vol. 30, No. 5, (May 2017) 785-790

of a hydro-kinetic turbine. In 2008, Garrett et al. studied maximum attainable energy by a fence of axial flow turbine [3]. In 2010, Yaakob et al. could design and test a Savonius vertical axis turbine for absorbing kinetic energy from very low speed (0.56 m/s) tidal flows [4, 5]. In 2012, Mehmood et al. could study many diffuser designs for tidal current turbines [6]. They showed that the kinetic energy in the diffuser throat can be increased with increasing diffuser length and diffuser angle of attack. During numerical studies the need to empirical models for modeling hydro-kinetic turbine has been clearly understood. In 2012 Shives and Crawford presented an empirical model and validated the Lawn model by repeating calculations [7]. In 2015, Ziaei et al. described manufacturing process of a microturbine. They emploed CNC machining for manufacturing a prototype of microturbine [8]. In 2016 a method for conceptual duct shape design for horizontal axis hydro-kinetic turbines was presented and extensive duct shapes were numerically studied [9]. The optimum duct shape has shown the greatest frictional drag coefficient and the minimum flow separation [9]. In the present paper method of design and manufacturing of a multi-diffuser axial hydro-kinetic turbine is discussed. A ducted turbine comprised of two high performance propellers has been manufactured that can be converted to many experimental models for studying various diffuser enhancing effects.

2. DESIGN AND MANUFACTURING OF MICRO-HYDRO-KINETIC TURBINE

Design of a hydro-kinetic turbine is not a straightforward process. Many analytical investigations, numerical analysis and experimental data may be required for finding optimum configuration of a ducted hydro-kinetic turbine. Knowledge from flow characteristics such as separation and energy transfer in the wake flow can determine the limits of the design variables. The maximum size of the duct can be predicted by analytical studies. As the turbine components increase optimization becomes more complicated. The nonlinearity of governing equations is another challenge. For solving this proplem each component of the turbine such as rotor, ducts and guides was optimized independently. Coupled CFD data with one-dimensional simulation of the flow field was employed for determining the optimum duct cross sectional areas. Turbine's rotor includes two propellers: A three-blade axial propeller with power coefficient close to 0.46 and a mixed propeller with six blades and higher power coefficient that is reached up to 0.73 at high speed flows. As shown in Figure 1, the propellers were manufactured with three-dimensional print method.

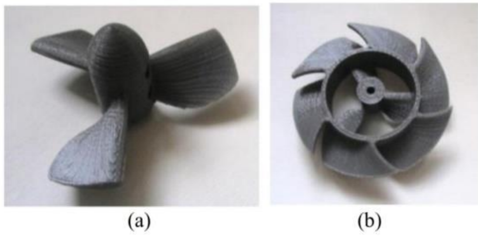


Figure 1. a) Three-blade axial propeller manufactured with PLA material and b) six-blade axial propeller with three-blade support inside it

For fastening duct parts screw pitches were employed. Assembling of the turbine is shown in Figure 2. With three-dimensional print method the holes diameter were shrinked up to 0.4 mm than CAD model. This problem was solved by defining reasonable tolerance for increasing diameter of pitch-holes.

3. EXPERIMENTS ON THE PROPELLER'S PERFORMANCE

Dynamic balance of propellers was investigated with high speed rotation tests in air and water. An active dynamo-motor was used for rotating the propeller with belt and pulley mechanism. A universal joint was employed for transfering mechanical power to the shaft as shown in Figure 3.

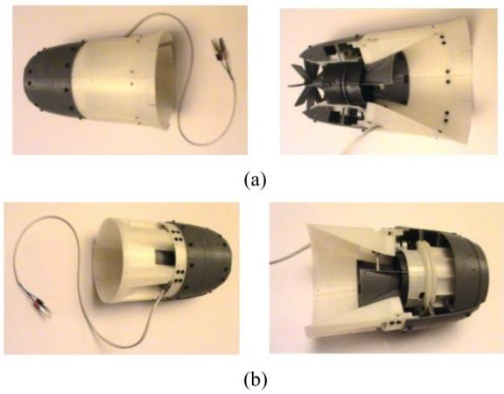


Figure 2. a) Assembly of the turbine with closed diffuser and b) assembly of ducted turbine with bypass diffuser

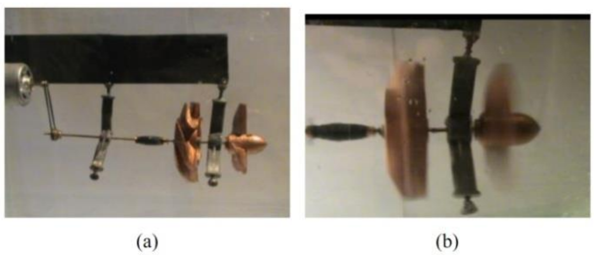


Figure 3. a) Test setup with a thin flexible slab that has black color and b) high speed rotation test in air at 3000 rpm

A thin flexible slab, made of hard plastic, was employed for observing vibrations. Forced vibration was observed at startup of the power transmission from the dynamo-motor. After speed-up of the propellers the slab vibration was damped very fast and the propeller worked without observable vibration.

A series of tests were done with constant input power for rotating propeller in quescent water. Where it was necessary the dynamo-motor was directly coupled to the rotor's shaft for highest power transmission. Based on manufacturing data and many tests, the efficiency of dynamo-motor was 84% and the maximum mechanical output power of dynamo-motor was found to be P_{mech} = 23.5 Watts. Power-supply in parallel mode was employed for producing controlled electrical current. The experimental results were summarized in Table 1. At the same flow condition CFD analysis data of ANSYS-FLUENT software were compared with experimental data.

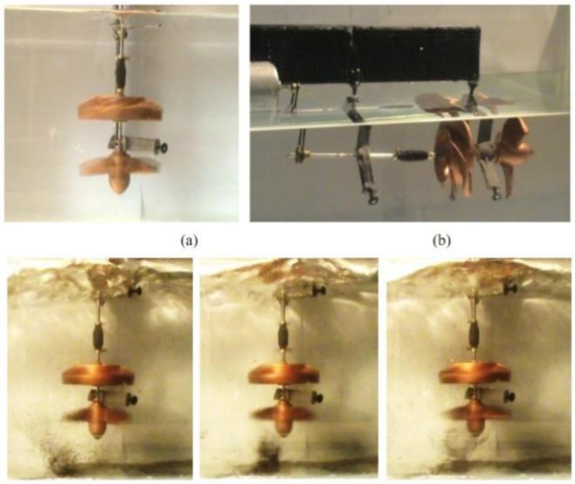
Based on flow field visualization neither sheet nor cloud cavitation were observed on the propeller blades. The rotation speed was between 0 to 754 rpm in clockwise direction. The maximum speed of propeller that dynamo-motor can speed it up was 754 rpm. For clockwise rotation the maximum propeller speed in water was approximately 30 percent lower than counter clockwise rotation. As a result, the propeller torque was higher during clockwise rotation. The clockwise rotation direction gives higher pressure at frontal faces of blades as is for an axial turbine.

Flow visualization in water with fine solid particles helps to see flow trajectories . The particles are initially accumulated in front of the propeller. In this experiment pressure side of blades are rear faces and the water level is rising above the propeller. At the bottom of the water reservior a swirling flow is induced. The induced swirling flow moves around propeller axis as shown in Figure 4-c.

Before testing the propellers in the water tunnel an experiment was done for calibrating load-cell. As illustrated in Figure 5-a the load-cell was accommodated in a vertical hydrofoil. The hydrofoil reduces excessive drag force. In this experiment a uniform wind flow field was generated with a 0.5 m diameter axial fan with angular speed of 1480 rpm. Experiment on axial propeller was also done in a high-speed water tunnel.

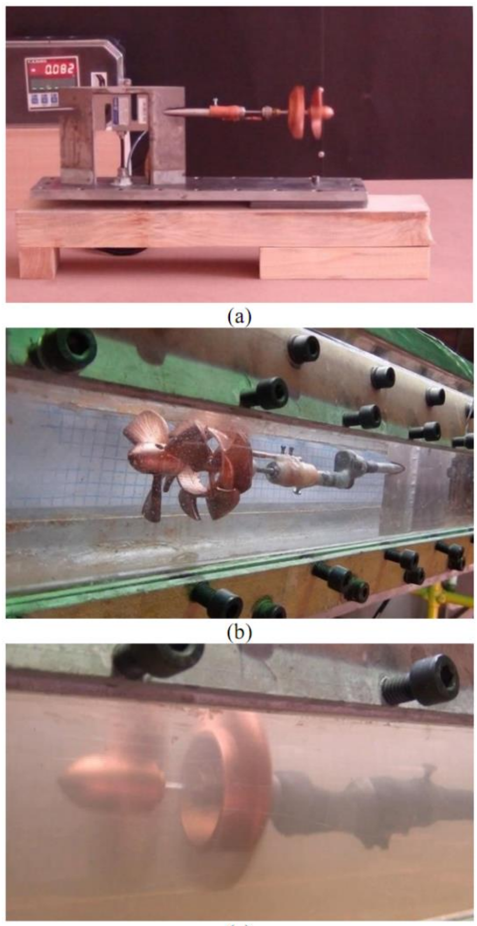
TABLE 1. Estimation of propeller torque in water with clockwise rotation

E, parallel (V)	I, parallel (A)	W (Watts)	ω (R.P.M), CW(+)/CCW(-)	Estimated T_0 (N.m)	T_0 from CFD simulations
(1.3+1.5)/2	2.78+2.73	7.714	+521	-0.120	-0.108
(2.20+2.5)/2	4.91+4.90	23.05	+754	-0.248	-0.202



(a) (b) (c)

Figure 4. a) Propellers test for observing cavitation in quecient water. The pressure side of propellers is in front, b) vibration test setup in water and c) flow visualization in water with fine solid particles



(a) (b) (c)

Figure 5. a) Calibration of disk-brake and load cell in air, b) experimental setup in a high speed water tunnel and c) a photograph of propeller during power absorption at 2.85 m/s water flow

The cross section area of the test section was 10 cm × 12 cm. The propeller shaft was hold with a micro-disk brake to generate optimum torque of 0.12 N.m for area-averaged flow speed in the range of 2.5 m/s to 4.0 m/s. The drag force on the propellers was measured with the calibrated load-cell in the wind flow. For protecting propellers and brake system against the sudden impact of high speed water flow a flexible mechanism (with infinitesimal deformation and high damping factor) was designed. The flexible support covers propeller shaft, micro-brake and load cell.

According to the data in Table 2, nearly constant drag force coefficient was measured while a constant resistant torque of 0.12 N.m were exerted on the propeller shaft.

4. NUMERICAL STUDIES

Following the discussion in section 2, numerical study of the flow field around ducted propeller could determine performance of hydro-kinetic turbines and optimum geometry. The main features and details of numerical methods used for coupled flow simulations and propeller dynamics were presented in this section.

4. 1. Transient Simulation of Propeller's Dynamics with One-dimensional Actuator-disc Method The fluid flow was governed with one-dimensional quasi-transient equations which were obtained after integration from three-dimensional continuity and momentum equations. The governing equations are similar to one-dimensional actuator-disc method represented in reference [10]. Experimental or CFD simulation data can be fed to one-dimensional governing equations as the boundary value of the integrals. Two approaches have been employed for solving dynamic equation of motion for propeller based on steady/unsteady CFD data: Forth-order Runge-Kutta method and analytical integral solution (boundary value problem within each time step). Details of analytical integral solution for solving a second-order differential equation were reported in the literature [11].

TABLE 2. Measured data for axial propeller in a high-speed water tunnel

Area-average flow velocity, u_0 (m/s)	Re_D	Holding torque (N.m)	Experiment in water tunnel		
			C_D	C_p	λ
2.5	417,900	0.12	1.02	0.62	2.82
3.0	501,500	0.12	1.03	0.72	4.69
3.5	585,000	0.12	1.03	0.67	5.94
4.0	669,000	0.12	1.03	0.40	4.59

As a boundary value problem the dynamic equation for rotor motion was solved incrementally in the Banach space with unique solution. Transient growth rate for time variable was calculated based on energy conservation principle to hold constant angular increments for capturing accelerating motions. The Hilbert-Schmidt hypothesis was used for generating orthonormal vectors for solving matrix equations that include spars coefficient matrix.

4. 2. CFD Simulations for Micro-hidro-kinetic Turbine Full three-dimensional CFD simulation in ANSYS-FLUENT software illustrates the turbine performance. A user-defined function was written for solving dynamic equation of motion for propeller in one-degree of freedom rotation. The UDF finds the shear stress tensor and the isotropic tensor of mechanical pressure on the propeller surface. Then surface tractions on the propeller surface are calculated from which the propeller thrust and torque are estimated. The grid was generated in ANSYS-MESHING software with 980,000 tetrahedral elements for a fluid sub-domain around the propeller and 1,630,000-1,740,000 tetrahedral elements for a fluid sub-domain around various ducts. The mixture multiphase model was employed. The time step size was in the range 0.002 to 0.0001 s. The pressure based solver with coupled scheme for pressure-velocity coupling was employed. The hydro-static pressure was defined using the gravity option and a reference pressure point. Flow trajectories with contour plots for Re_D = 501,500 are illustrated in Figure 6. The Reynolds number was calculated based on mean diameter of hydro-kinetic turbine, D = 0.168 m and free stream velocity of 3.0 m/s. The turbulence intensity is higher at the position of strong vortices. The throat flow after propellers is a low energy flow while hydro-static pressure gradient has turned the throat flow to upper side of the diffuser.

4. 3. Numerical Simulation Results The CFD simulation data for the turbine configuration in Figure 2-a, are summarized in Table 3. The inlet and exit areas of the duct were changed to study the influence of duct cross sectional area on the absorbed power.

The CFD simulation data for the turbine configuration with open bypass channels in Figure 2-b are summarized in Table 4.

The diffuser with bypass channels (Figure 2-b) can increase throat velocity in a transient flow. It has better transient characteristics for a tidal turbine than a single diffuser. However the steady performance of a diffuser was better up to few percent. Dynamic simulation of propellers using CFD data has determined transient angular speed of propeller (Figure 7) for optimum ducted turbine (Figure 2-a).

At the start of three-dimensional CFD simulation, the transient absorbed power of hydro-kinetic turbine exhibits unphysical oscillations with large amplitude. This behaviour is illustrated in Figure 8. The one-dimensional solution has the advantage of better stability for prediction of transient power absorption.

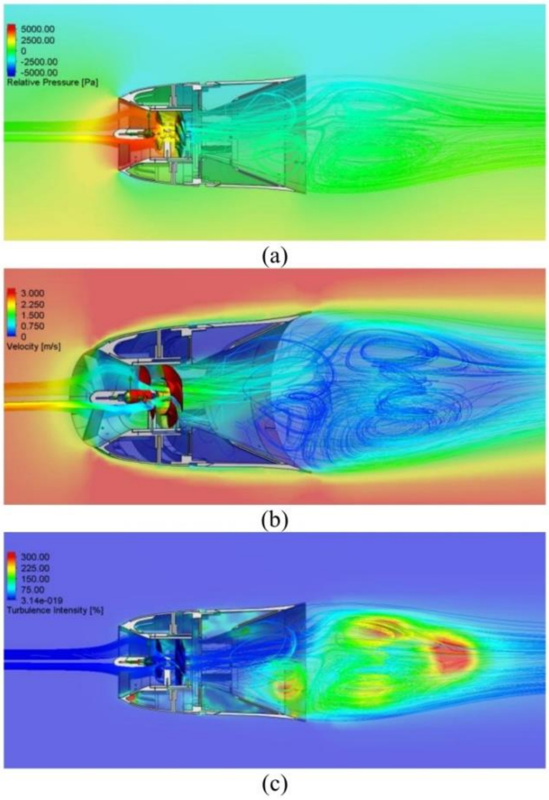


Figure 6. Graphical results and path-lines for optimum ducted turbine at 3 m/s water flow. a) Relative pressure, b) velocity magnitude and c) turbulence intensity

TABLE 3. Numerical simulation results for hydro-kinetic turbine with closed diffuser (Figure 2-a)

	Case-study	**1**	**2**	**3**	**4**	**5**
Solution method	A_i/A_{th}	2.3	4.3	6.3	2.3	2.3
	A_e/A_{th}	4.3	4.3	4.3	6.3	8.3
Three-dimensional transient solution with ANSYS-FLUENT software	ω (R.P.M)	1477	1560	1592	1655	1840
	P (Watts)	61.86	65.34	66.68	69.32	77.07
	C_P	0.52	0.55	0.56	0.58	0.65
One-dimensional actuator-disc method with CFD data	ω (R.P.M)	1489	1491	1494	1531	1569
	P (Watts)	62.37	62.45	62.58	64.12	65.72
	C_P	0.52	0.52	0.53	0.54	0.55

TABLE 4. CFD data of ANSYS-FLUENT software for ducted turbine with bypass channels in Figure 2-b

	Case-study	**1**	**2**	**3**	**4**	**5**
Solution method	A_i/A_{th}	2.3	4.3	6.3	2.3	2.3
	A_e/A_{th}	4.3	4.3	4.3	6.3	8.3
Three-dimensional transient solution with ANSYS-FLUENT software	ω (R.P.M)	1480	1576	1624	1635	1737
	P (Watts)	61.98	65.99	68.02	68.48	72.68
	C_P	0.52	0.56	0.57	0.57	0.61

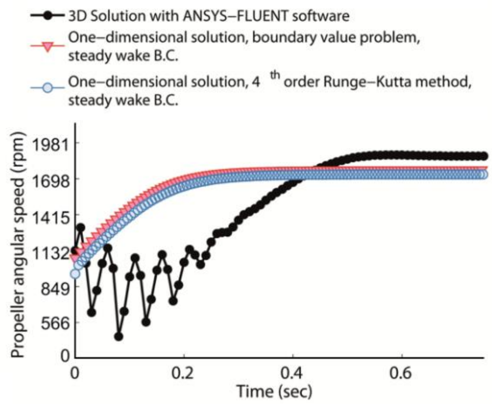


Figure 7. Transient propeller angular speed for optimized hydro-kinetic turbine (Figure 2-a) with single diffuser

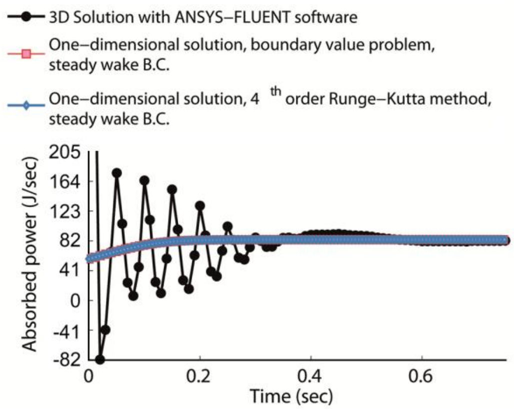


Figure 8. Transient absorbed power for optimized hydro-kinetic turbine in Figure 2-a with a single diffuser

5. CONCLUDING REMARKS

Design, fabrication and dynamic simulation of a micro-hydro-kinetic turbine were discussed and many experiments were done for illustrating the propeller performance. The most important concluding remarks for the present study are:

- Centering of propellers and bearings yields a dynamically balanced rotor for high speed rotations up to 3000 rpm. The propellers also could preserve dynamic balance characteristics in water.
- At high speed rotation cavitation was not observed around propellers.
- The performance of propellers was studied with both experimental study and CFD simulations. CFD data were in agreement with the experimental measurements. On this basis, the presented CFD method for simulation of fluid flow over a ducted turbine is valid.
- Most parts of ducts were designed as thin shells. This is a successful technique for lowering cost and material during 3-D printing. However structure of shells have been strengthened with stiffener and supports.
- A substantial reduction in power coefficients of ducted turbine was for neglecting optimum flow conditions. The CFD simulation data were not coincident with optimum power coefficient of ducted turbine. The optimum power coefficient of the ducted turbine was 30% higher than 0.73 corresponding to un-ducted propellers.

6. REFERENCES

1. Islam, Q. and Chandra Mandal, A., "A studyof the design of horizontal axis wind turbine", ***International Journal of Engineering***, Vol. 6, No. 2&3, (1993), 117-124.
2. Setoguchi, T., Shiomi, N. and Kaneko, K., "Development of two-way diffuser for fluid energy conversion system", ***Renewable Energy***, Vol. 29, No. 10, (2004), 1757-1771.
3. Garrett, C. and Cummins, P., "Limits to tidal current power", ***Renewable Energy***, Vol. 33, No. 11, (2008), 2485-2490.
4. Yaakob, O.B., Tawi, K. and Sunanto, D.S., "Computer simulation studies on the effect overlap ratio for savonius type vertical axis marine current turbine", ***International Journal of Engineering-Transactions A Basics***, Vol. 23, (2010), 79-88.
5. Yaakob, O., Suprayogi, D., Ghani, M.A. and Tawi, K., "Experimental studies on savonius-type vertical axis turbine for low marine current velocity", ***International Journal of Engineering-Transactions A: Basics***, Vol. 26, No. 1, (2012), 91-98.
6. Mehmooda, N., Lianga, Z. and Khanb, J., "Study of naca 0015 for diffuser design in tidal current turbine applications", ***IJE Transactions C: Aspects***, Vol. 25, No. 4, (2012), 373-380.
7. Shives, M. and Crawford, C., "Developing an empirical model for ducted tidal turbine performance using numerical simulation results", ***Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy***, Vol. 226, No. 1, (2012), 112-125.
8. Tabatabaei, S.Z., Hashemi, A. and Meysami, A., "The manufacturing process of a 100-kw prototype microturbine as a distributed generation method in iran (technical note)", ***International Journal of Engineering-Transactions A: Basics***, Vol. 28, No. 1, (2014), 145-153.
9. Zahedi Nejad, A., Rad, M. and Khayat, M., "Conceptual duct shape design for horizontal-axis hydrokinetic turbines", ***Scientia Iranica. Transaction B, Mechanical Engineering***, Vol. 23, No. 5, (2016), 2113.
10. Houlsby, G., Draper, S. and Oldfield, M., "Application of linear momentum actuator disc theory to open channel flow", ***Report no. OUEL***, Vol. 2296, No. 08, (2008).
11. Mehri, B. and Nejad, A.Z., "Application of the singular boundary value problem for investigation of piston dynamics under polytropic expansion process", ***International Journal of Mathematical Modelling & Computations***, Vol. 2, No. 3, (2013).

Numerical and Experimental Investigations for Design of a High Performance Micro-hydro-kinetic Turbine

A. Zahedi Nejad[a], M. Rad[b], M. Khayat[a]

[a] *Department of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran*
[b] *Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran*

PAPER INFO

Paper history:
Received 05 June 2016
Received in revised form 07 February 2017
Accepted 10 February 2017

Keywords:
Duct
Dynamic Simulation
Fabrication
Hydro-kinetic Turbine
Propeller Test

چکیده

در مقاله حاضر طراحی و ساخت یک توربین آبی کوچک انرژی-جنبشی با عملکرد بالا مورد بحث قرار گرفته است. هدف اصلی ساخت یک مدل آزمایشی تجهیز شده از توربین آبی انرژی-جنبشی است که بتواند بیشترین انرژی را از جریان آب جذب کند. یک توربین چند شکلی دارای داکت شامل یک دیفیوزر چند قطعه ای ساخته شده است که می تواند به چندین مدل آزمایشی برای بررسی تأثیرات افزایش قدرت دیفیوزر تبدیل گردد. روتور توربین شامل یک پروانه محوری سه پره ای و یک پروانه ترکیبی شش پره ای با ضریب توان بالا است. برای مشاهده جریان، اندازه گیری گشتاور و مشخص کردن تعادل دینامیکی در سرعت چرخش بالا در آب و هوا آزمایش های ساده ای روی پروانه انجام شده است. اطلاعات مهم برای طراحی و ساخت داکت و اجزای روتور که منجر به ساختار ایمن و متعادل روتور در سرعت چرخش بالا شدند مورد بحث قرار گرفتند. برای شبیه سازی دینامیکی توربین یک تابع که توسط کاربر تعریف شده برای نرم افزار ANSYS-FLUENT توسعه یافته است که اطلاعات انتگرالی را جمع آوری می کند و معادله دینامیکی روتور را در حرکت یک درجه آزادی حل می کند. چندین روش پایدار شبیه سازی دینامیکی برای کوپل شدن با جریان یک بعدی زمانمند در اطراف پروانه یک درجه آزادی پیشنهاد شدند و نتایج عددی در مقابل داده های شبیه سازی کامل دینامیک سیالات محاسباتی تأیید اعتبار شدند.

***doi**: 10.5829/idosi.ije.2017.30.05b.19*