Analysis and Design of a Submarine Low-Flow Horizontal-Axis Energy Harvesting System

Phuc Huu Nguyen*[1] and Tu Ngoc Nguyen*

Abstract – The paper presents the analysis and design of a low- flow horizontal- axis submarine turbine for energy harvesting technology aligned with the 21st century challenge of mitigating the effects of carbon emission on global warming and climate change in electricity production. The introduced technology is portable, has a compact modular design that makes assembly easy, uses off-the-shelf components to produce power. The modular and simple architecture of snap-and-fit components combined with the least number of both active and passive electrical components results in a highly efficient system for harvesting water energy. As the horizontal axis turbine is very much depending on water flow speed and depth, so a big challenge really exists when designing a turbine- generator operating in a low- flow environment. Therefore, some modifications must be made on the current speed or the turbine by introducing a Venturitube to enable the turbine to work properly in low current speed. In the paper the impeller blade profile calculation is carried out using the simple vortex panel method (VPM) first and Computational Fluid Dynamics (CFD) softwares then are used in verifying the design. Gambit and Fluent softwares are used for blade mesh generation and fluid dynamic performance simulation, respectively. The project target is to design a turbine- generator capable of producing 1 kW 50-Hz 220 VAC power at a depth of 2 m, flow velocity of 1.3 m/s. Comparison is made for performances between VPM- based design and CFD simulation, in terms of lift force, drag force coefficients and flow velocity around the profile. The calculation results, which sufficiently match those obtained from CFD softwares, show the effectiveness of the panel method analysis in use.

Keywords – energy harvesting, kinetic energy, low- flow turbine-generator system, turbine impeller, vortex panel method.

1. INTRODUCTION

The indiscriminate current use of conventional hydropower technology has led to the construction of dams that seriously affect and damage the ecology and biodiversity of vast regions. Therefore, the current trend is to develop energy sources beneficial to society in all respects.The idea of offering this technology is based on a basic fact that any moving mass is inherently storing kinetic energy. This fact allows generators to extract part of the water energy without causing environmental destructions. With the modern manufacturing techniques the production of plastic mold with high reliability can mimic and realize a variety of complicated shapes, when they are used in sensitive environments with natural appearance. In the system, the impeller is the main component in charge of converting flow kinetic energy into mechanical to drive the generator rotor.

Designing impeller blade profile is a classical problem in turbine design, and there are few methods in use, such as: vortex panel method, lift method, conformal mapping method. In the paper the vortex panel method is applied because of its simplicity, conveniences on computer programming, while ensuring acceptably good results, compared with those obtained from more sophisticated methods of commercial softwares.

*Ho Chi Minh City University of Technology, 268 Ly Thuong Kiet, Dist 10, Ho Chi Minh City, Vietnam.

[1] Corresponding author; Tel: 848 913 846 596.
Email: nhphuc@hcmut.edu.vn.

2. LOW- FLOW GENERATION SYSTEM

There is a large body of research information concerning how to compute specific speed for conventional, large hydropower plant design [1]-[3], which presents a review of research progress in horizontal-axis current turbines, shows that the majority of research has started with the ultimate aim of installing a real large/medium- scale current energy extraction device. However, for micro-hydro there is not much information regarding how to design a complete system of low-head/low-flow horizontal-axis turbines. It is remarkable in [4] and [9], from the works of Mohammed *et al.* and Baylar and Ozkanm, the application of Venturi principle to water aeration systems was explored and analysis was made on the effect of the addition of a duct on the hydrodynamic performance of the turbine. The numerical results from these works show that the addition of duct improves significantly the efficiency of low- power turbine.

Operating principle and components of the system

The turbine is a molded Venturi shape inside a rectangular shaped outer structure to provide the required flow characteristics for power harvesting, and the structural stability required in its installation, respectively [5], [8], [10], [11]. Both flow-rate and volumetric discharge at the Venturi throat are optimized to produce the maximum angular torque on a specially designed impeller comprising of metal and plastic in a single molding. It is important for the impeller mass and the mass distribution along the radial length of the impeller to be optimized such that maximum angular torque can be produced relative to the desired discharge at the throat of the Venturi. The distributed mass causes maximum centripetal force to overcome fluid friction. For this purpose a water repellant plastic molding will be targeted. As water enters the Venturi inlet, the narrowing gap between the Venturi wall and the submarine shell causes the velocity of water to increase and impinge on the impeller at a higher velocity,as shown in[5], [10], [11] (Figure 1).

For hydrokinetic generators, the power output increases as a cube of the velocity. Guide vanes that are appropriately angled provide a means to spiral the water entering the Venturi such that additional torque can be generated by the water impinging upon the impeller. A variable speed permanent magnet alternator (PMA) rotor is coupled to an impeller and housed inside a watertight submarine-shaped shell. An oil-sealed bearing preserves a watertight seal and allows the axially mounted impeller to turn the rotor shaft of the generator. An AC-DC-AC power converter that is placed in an external environment (typically in the control box adjacent to the service transformer) converts the AC power produced by the variable-speed alternator to nominal voltage at rated frequency, and delivers power at the desired nominal frequency.

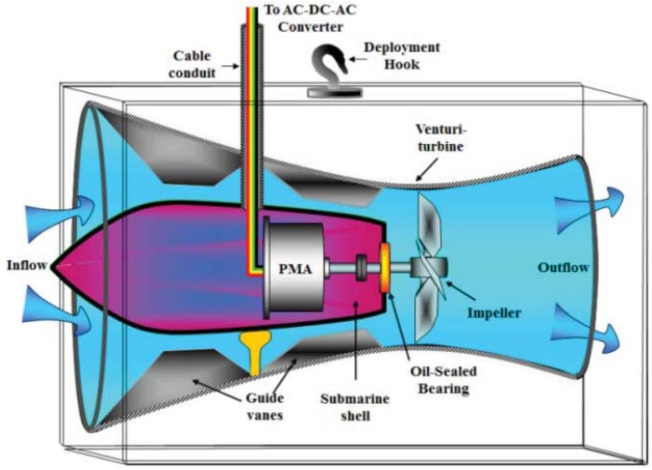

Fig. 1. Low- flow energy harvester.

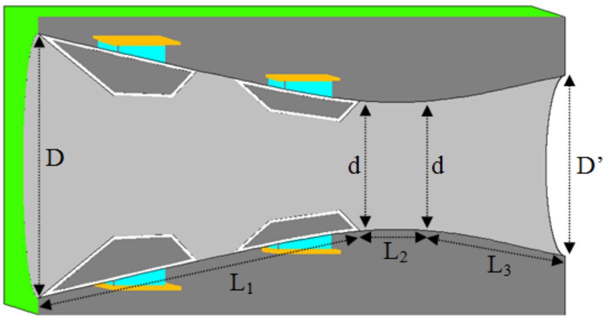

Fig. 2. Harvester dimensions.

This feature induces better rotational motion on the impeller and hence acts to improve the turbine energy conversion ratio, which is the hydraulic energy converted to mechanical energy divided by available energy. An oil-filled watertight bearing provides isolation from water outside the submarine and electrical components inside the plastic molding of the submarine. A cable conduit connected by awatertight sealing to the submarine shell allows electrical phase wiring to be extracted from the power converter output terminals and connected externally to transformers, loads and the local electric utility.

3. DESIGN SPECIFICATIONS

The project is to design a low- power generation system capable of producing 1 KW electrical power at 220 VAC, 50 Hz, single phase at a depth of 2 m, inlet flow velocity of 1.3 m/s.

The power of water flowing through a constant cross section of *A* is determined by:

$$P = \frac{1}{2}\rho A V^3 \text{ (W)},$$

where ρ: water density, V: flow velocity

Turbine hydraulic head is determined from the following Bernoulli equation:

$$H = Z_0 - Z_1 + \frac{p_0 - p_1}{\gamma} + \frac{V_0^2 - V_1^2}{2g}$$

The system is installed below the surface with Zo = 2.0 m. As the depth from the water surface to the harvester centerline is short, so the difference in terms of pressure, velocity at the two mentioned positions is negligible, the turbine hydraulic head therefore is approximately 2 m.

The following dimensions are shown in Figure 2 for design and calculation of the harvester components: inlet diameter D= 1.5 m, Venturi throat diameter d = 1.05 m, discharge outlet diameter D'= 1.15 m, L1= 1.2 m, L2= 0.2 m, L3= 0.8 m.

Power Generation Sizing: the calculated flow velocity entering the impeller blades is 2.66 m/s, and the kinetic power delivered at the Venturi throat is 8078 W. With the entering flow velocity of 2.66 m/s at the impeller with diameter of 1 m and the turbine shaft being directly coupled to the generator rotor, the rotor speed will be 50 rev/min with an output torque of 259 Nm. The expected power output of the generator is 1000 W.

4. DESIGN OF TURBINE IMPELLER BLADES USING VORTEX PANEL DISTRIBUTION METHOD

With an incompressible flow of constant density flow, i.e. ρ = constant, the volume of the fluid element is also constant. $\nabla\vec{V}$ is the time rate of change of the volume of a fluid element, per unit volume, the following equation holds true: $\nabla\vec{V} = 0$. For an irrotational flow the velocity can be expressed as the gradient of a scalar function called the velocity potential denoted by Φ, $\vec{V} = \nabla\Phi$ and the Laplace's equation follows: $\nabla^2\Phi = 0$ [6]. As Laplace's equation is linear, hence any number of particular solutions to the equation can be added together to obtain another solution. This establishes a basic philosophy of the solution of incompressible flows, namely, that a complicated flow pattern for an irrotational, incompressible flow can be synthesized by adding together a number of elementary flows which are also irrotational and incompressible [6].

Vortex panel method is a numerical method used to design the camber line of impeller blades. In a forward problem, a complicated pattern for an irrotational, incompressible flow can be synthesized by adding together a number of elementary flows, such as uniform flow, source flow or vortex flow. In uniform flow with velocity V∞ moving in the *x*-direction, $\Phi = V_\infty x$. In source flow with source strength Λ, $\Phi = -\frac{\Lambda}{2\pi}\ln r$, whereas in vortex flow with vortex strength Γ, $\Phi = -\frac{\Gamma}{2\pi}\theta$, where the vortex strength Γ is the circulation around the vortex filament and the circulation is defined as: $\Gamma = -\int \vec{V}.\vec{dS}$.

In the forward problem, the streamline flow around a given blade profile can be determined by adding together the flow results obtained from analysis of individual elementary flows, such as uniform flow and defined vortex distribution $\gamma(s)$ over a series of panels distributed along the blade profile. In the design problem, a reverse approach is treated where uniform flow and vortex distribution are determined from given data, and the blade camber line geometry is to be determined. In [7] the vortex distribution can be expressed by means of a trigonometric series of coefficients A_o, A_1, A_2,…. These coefficients can be determined from given data of hydraulic head H, lift force P_s and other parameters related to the blades. To build the blade profile of finite thickness the vortex panel method is first used to generate the camber line of blade profile, then high quality aerodynamic profiles are applied to generate other thicknesses at different blade sections. The vortex distribution in use must satisfy the condition of the circulation of flow around the blade profile equal to the sum of vortex strengths $\gamma(s)$: $\Gamma = \int_{-L/2}^{L/2}\gamma(s)\,ds$, where L: blade profile chord.

Vortex distribution can be expressed as a trigonometric series:

$\Gamma(\theta) = A_0 cotg\ \theta/2 + A_1 sin\ \theta + A_2 sin\ 2\theta + A_n sin(n\theta)$, in which $A_0, A_1, A_2, \ldots A_n$ are coefficient of the series.

And the source distribution must satisfy the condition of the total flows of sources on the camber line must be zero: $\int_{-L/2}^{L/2} q(s)\,ds = 0$, in which $q(s)$ can be expressed as a polynomial, generally of 3rd order: $q(s) = B_1 + B_2 s + B_3 s^2 + B_4 s^3$

The combination of elementary flows of uniform, source and vortex types will be a closed curve, which is the profile determined.

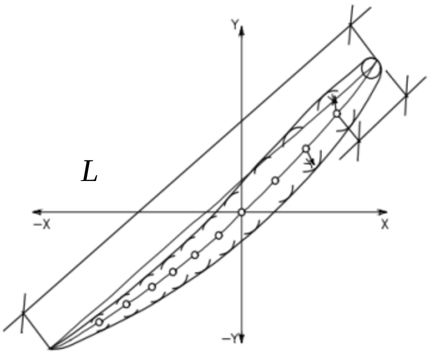

Fig. 3. Designed camber line with added thicknesses.

Based on the approach and formula and graphs from empirical studies given in [7] *x-y* coordinates of the blade camber line can be calculated in the first iteration, in which the camber line is the infinitely thin profile determined. In the second iteration, the vortex distribution will be placed on the first camber line calculated, then the calculation repeats. Generally, two iterations are sufficient with acceptable accuracy. Finally, to obtain a profile with thickness some sample profiles with good energy features and resistance to cavitation effects are used to obtain the thickness corresponding to each section along the camber line. The envelope of the thicknesses thus designed will form the final profile of the impeller blade (Figure 3).

Following parameters are used for impeller design: harvester head H= 2 m, revolutions per minute n= 50, hub-to-tip ratio d_b/D_1=0.3, number of blades Z_1= 6, maximal thickness δ_{max} at profile base = 12%, minimal thickness at blade outermost tip δ_{min} = 4%, linear relationship between thickness along blade radius $\delta = f(r)$ being assumed, L/t coefficient of 1.45 at profile base and of 1.16 at outermost tip (t: length between 2 consecutive blades). Different impeller blade profiles I, II, III, IV, V are considered, with profiles I to V denoting profiles going from the outermost blade tip to the blade base, at different lengths from r= 0.45m to 0.25 m, respectively (Figure 4). Corresponding *x-y* coordinates of the camber line of each profile are calculated and shown in Tables 1 to 5 and in Figures 5 to 9. Table 6 shows blade profile parameters and corresponding lift coefficient C_l and drag coefficient C_d calculated.

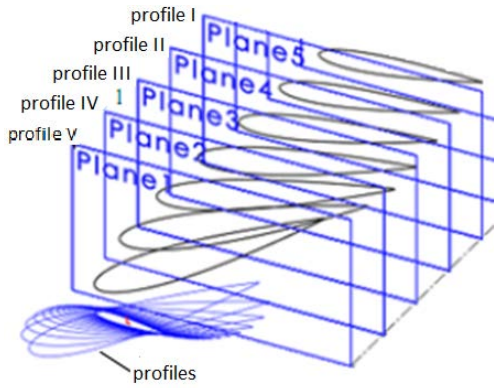

Fig. 4. Blade profiles I to V at different radius lengths.

Table 1. (*x,y*) coordinates of the camber line of profile I.

x(m). 10^{-3}	-8	-3	-1.5	0	4	11	21.2
y(m). 10^{-3}	-290	-193	-97	0	97	193	290

Table 2. (*x,y*) coordinates of the camber line of profile II.

x(m). 10^{-3}	-20	-18	-10.5	0	16.1	35	53.6
y(m). 10^{-3}	-275	-183	-91.6	0	90	180	270

Table 3. (*x,y*) coordinates of the camber line of profile III.

x(m). 10^{-3}	-49	-35	-18	0	20.9	44.7	71.3
y(m). 10^{-3}	-250	-166	-83	0	82.3	163.8	244.4

Table 4. (*x,y*) coordinates of the camber line of profile IV.

x(m). 10^{-3}	-62	-44	-23.7	0	27.2	58.4	94.1
y(m). 10^{-3}	-22.3	-148	-73.6	0	72.3	143	211.5

Table 5. (*x,y*) coordinates of the camber line of profile V.

x(m). 10^{-3}	-76	-53	-28	0	30.9	65.2	103.4
y(m). 10^{-3}	-186	-123	-61	0	59.6	117.3	172.5

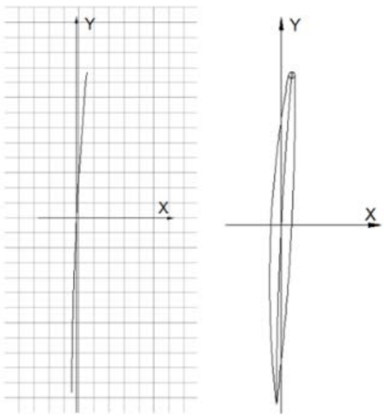

Fig. 5. Camber line of profiles I.

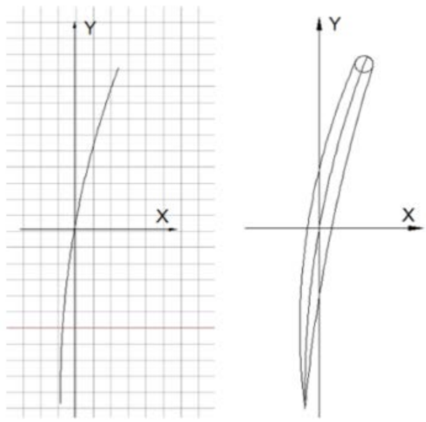

Fig. 6. Camber line of profile II.

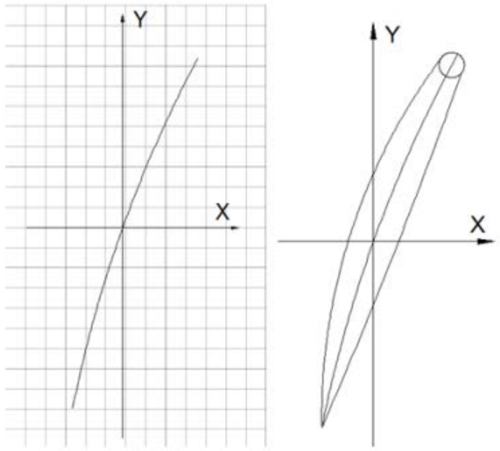

Fig. 7. Camber line of profile III.

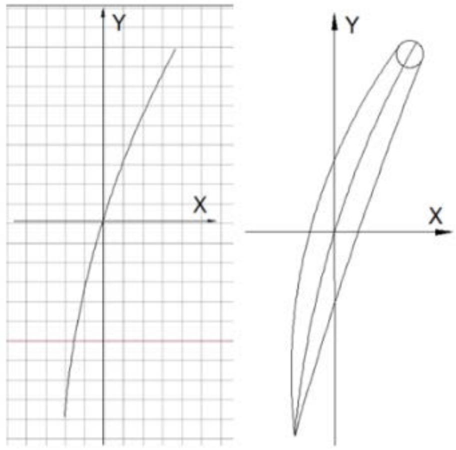

Fig. 8. Camber line of profile IV.

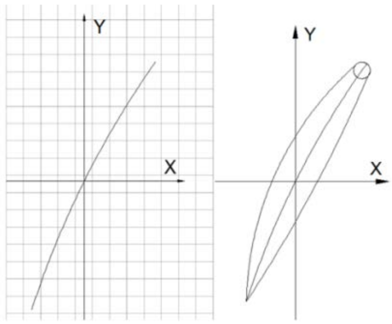

Fig. 9. Camber line of profile V.

Table 6. Blade profiles I to V and corresponding parameters.

No	Parameters	Blade profile				
		I	II	III	IV	V
1	*r, (m)*	0,45	0,40	0,35	0,30	0,25
2	$t = 2.\pi.r / z_1(m)$	0,50	0,45	0,39	0,38	0,27
3	*L = t.L/t (m)*	0,58	0,55	0,51	0,46	0,40
4	β_c	86°5’	79°27’	71°8’	61°17’	51°23’
5	δ_{max} %	4,0	6,0	8,0	10	12
6	$C_l = \frac{2\Gamma}{LV_\infty}\eta_s.K$	1,14	1,00	0,85	0,75	0,66
7	C_d	0,102	0,101	0,09	0,073	0,052

5. CFD FLOW SIMULATION

Computational Fluid Dynamics (CFD) softwares Gambit and Fluent are used for blade profile mesh generation and flow simulation, respectively. Simulations of different profiles I, II, III, IV and V were carried out. The simulation results also show the lift coefficient C_l increasing from the base profile V(C_l= 0.4) to the outermost profile I (C_l= 0.6), compared with C_l of 0.66 and 1.14,obtained from panel method design, respectively,at corresponding profiles. The simulated drag coefficient C_d increases from the base profile V (C_d= 0.08) to the outermost profileI (C_d= 0.11), compared with calculated C_d of 0.052 and 0.102, respectively, at corresponding profiles. In Figures 10 and 11, for a typical simulation at the profile V (at the blade base of radius length r = 0.25 m), meshes are generated and flow velocity distribution pattern around the blade are shown. A maximum flow velocity of 2.42 m/s is obtained from the numerical simulation; this fairly well meets the velocity of 2.66 m/s from previous calculation, corresponding to 50 revolutions per minute as required by the generator.

In relation to C_d the discrepancy between the simulated and calculated results is of relative magnitude error of 7%- 35%, while those for C_l are in the range of 39%- 47%.Maximal flow velocity of 2.42 m/s is obtained from numerical simulation, whereas that of 2.66 m/s is from calculation, and the corresponding relative amplitude error is of 9%, which is not unreasonable. The above error analysis therefore has shown that the panel method model can be effectively used for a preliminary design with sufficient accuracy to warrant its being employed further to get refined results with more advanced CFD softwares.

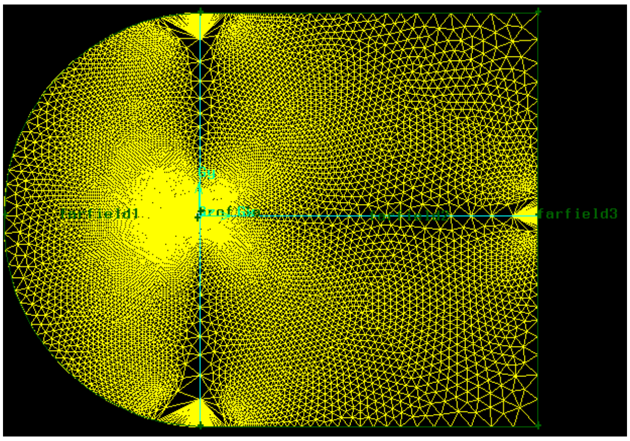

Fig. 10. Mesh generation for blade profile.

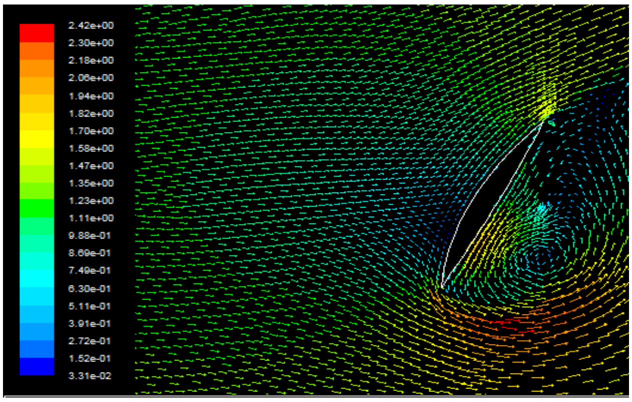

Fig. 11. Flow velocity pattern around blade profile.

6. CONCLUSION

In the paper a turbine-generator system with Venturi tube was designed and analyzed by applying the simple panel method with assumptions of incompressible and irrotational flow. Different impeller blade profiles are designed and verified by CFD softwares. The calculation results being acceptably similar to those obtained from numerical simulation prove the validity of the method proposed. Further refined calculation and comparisons with lift and conformal mapping methods, as well as with experimental setups will be carried out in the future to improve the method.

REFERENCES

[1] Schweiger F. and J. Gregory. 1987. Developments in the design of Kaplan turbines. *Water Power & Dam Construction* 39(11): 16-20.

[2] Sadek R. and M.A. Sinbel. 1960. Water turbines and dimensional analysis. *Water Power* 12(10): 381-389.

[3] Ng K.-W., Lam W.-H., and Ng K.-C., 2013. 2002–2012: 10 years of research progress in horizontal-axis marine current turbines. *Energies 6:* 1497-1526*; doi: 10.3390/en6031497.*

[4] Mohammed M., Tarfaoui M., and Laurens J.M., 2014. Predictions of the hydrodynamic performance of horizontal axis marine current turbines using a panel method program. In Proceedings of *International Conference on Renewable Energies and Power Quality (ICREPQ'14)* Cordoba (Spain), 8th to 10th April, 2014 Renewable Energy and Power Quality Journal (RE&PQJ) ISSN 2172-038 X, No.12, April 2014.

[5] Prasad R. and P.H. Nguyen. 2012. Hydropower energy recovery (Hyper) from water-flow systems in Vietnam. In *Proceedings of the 10th IEEE International Power and Energy Conference* (*IPEC 2012-Ho Chi Minh City).* Ho Chi Minh City, December 12-13, 2012.

[6] Wendt J.F. (Ed.) 2009. An Introduction to Computational Fluid Dynamics. *Third Edition, Springer-Verlag.*

[7] Huỳnh V.S., 2004. Design of Water Turbine Impellers. In *Water Turbines* by Huynh V.S. and Nguyen T.X.T., eds. Science & Technology Editions, chapter 5, pp: 59-116.

[8] Prasad N.R. and S.J. Ranade. 2014. Hydropower Energy Resource (HyPER) Harvester; *Department of Energy 2014 Water Power Program Peer Review Compiled Presentations - HydroPower Technologies, Washington Feb 25-28, 2014.*

[9] Baylar A. and F. Ozkanm. 2015. Application of Venturi Principle to Water Aeration Systems; *Environmental Fluid Mechanics. Available at: http://link.springer.com/content/pdf/10.1007%2Fs10652-005-5664-9.pdf#page-1*.

[10] Prasad N.R., Ranade S.J., and Nguyen P.H., 2015. Low-head hydropower energy resource harvesting: analysis and design of a Venturi turbine. In the *Proceedings of the International Symposium on Electrical- Electronics Engineering (ISEE 2015), HoChi Minh City Oct 3rd- 2015; pp 254-263.*

[11] Prasad N.R., Ranade S.J., and Nguyen P.H., 2015. Low-head hydropower energy resource harvesting: design and manufacturing of the (HyPER) harvester. In the *Proceedings of the International Symposium on Electrical- Electronics Engineering (ISEE 2015), HoChi Minh City Oct 3rd- 2015; pp 262-270.*