Proceedings of the 2013 ASME International Mechanical Engineering Congress and Exposition
IMECE2013
November 15-21, 2013, San Diego, California, USA

IMECE2013-66417

DESIGN AND PERFORMANCE TESTING OF A DUCTED SAVONIUS TURBINE FOR MARINE CURRENT ENERGY EXTRACTION

Jai N Goundar
The University of the South Pacific
Suva, Fiji
goundar_j@usp.ac.fj

Deepak Prasad
The University of the South Pacific
Suva, Fiji
prasad_dp@usp.ac.fj

Mohammed Rafiuddin Ahmed
The University of the South Pacific
Suva, Fiji
ahmed_r@usp.ac.fj

ABSTRACT

Marine current energy is a reliable and clean source of energy. Several marine current turbines have been developed over the years, most of the turbines perform well at velocities over 2 m/s and need to be installed at depths of 20 - 40 m. Placing an appropriately designed duct or shroud around the turbine significantly improves the turbine's performance. Ducted Savonius turbines can operate at low depths, since large clearance is not required because turbulent flow has little effect on the performance of the Savonius rotor. Ducted Savonius turbine has simple components and can be easily fabricated in Pacific Island Countries (PIC) and other places that do not have advanced manufacturing industries. A ducted Savonius turbine was designed for a location in Fiji, to operate at a rated marine current speed of 1.15 m/s and cut in speed of 0.2 m/s. The model of ducted Savonius turbine, scaled down to 1/20, was fabricated and tested in a water stream with a velocity of 0.6 m/s and was validated with commercial Computational Fluid Dynamics (CFD) code ANSYS-CFX. Finally, a full scale numerical model was constructed to study the flow characteristics and compute the performance. The area ratio of the duct of 2.5:1 (inlet to turbine section) shows significant increase in kinetic energy and an improved turbine performance. The maximum efficiency of the turbine is around 50% at a tip speed ratio (TSR) of 3.5 and the maximum power produced is 10 kW at the rated speed of 1.15 m/s and 63.4 kW at a free-stream velocity of 2.15 m/s.

INTRODUCTION

Pacific Island countries (PICs) like Fiji burn about 70900 tonnes of fuel yearly to produce electricity [1] and the cost of fossil fuels is increasing because the amount of fossil fuel that was easily accessible has decreased. The fuel cost further increases when it is transported to PICs. The demand for electricity keeps on increasing due to increase in population, increase in manufacturing industries and industries moving from manual to automated processing. Burning fossil fuels emits greenhouse gases causing global warming and other environmental problems. The electricity from the main grid does not reach to small isolated islands and some parts of rural areas. Diesel generators are used at these places, but it is not a good solution, it costs even high to transport fuel to these places. To minimize the use of fossil fuels to produce electricity, PICs have now focused their attention on renewable energy to generate electricity; renewable energy resources are clean energy sources and have minimum effect on the environment. The Fiji electricity authority (FEA) is targeting to generate 90% of energy through renewable resources by 2015 [1].

Many renewable energy resources have been exploited over the years, some of the renewable energy resources that are available in the form of mechanical energy and could be tapped just by placing a mechanical energy conversion device (a turbine) are hydropower, wind energy, wave energy, tidal energy, and marine current energy. Due to limited resources available on land, ocean energy appears a good option; ocean covers more area than land in PICs.

Many marine current convertors have been developed over the years. Most commonly used marine current turbine is horizontal axis marine current turbine (HAMCT); it has maximum efficiency of 45 - 48 % [2-5]. However, for HAMCT or any other marine current converter which is governed by lift, its performance significantly deteriorates at high turbulence levels and when blade fouling occurs [6]. The blade needs to be regularly cleaned and maintained to avoid losses. The manufacturing cost of this type of turbine is usually high because of its complex blade geometry and active blade controls that are required for it to perform well at varying operating conditions. The major problem with lift governed turbines is cavitation; the local pressure on blade surface significantly drops causing cavitation incepting which damages the turbine blades. To overcome these difficulties, drag governed turbines could be used to tap marine current energy. A common drag turbine that is used in wind turbine technology is Savonius rotor; which can also be used as hydrokinetic turbine. It is very cheap and simple to manufacture and can operate at low velocities. Several Savonius rotors have been designed and used to extract wind energy, some are presented in refs. [7-10] which have efficiencies of 15% – 21%. Mohamed et al. [11] reported that Savonius rotor efficiency could be increases by 27%, by placing an obstacle before turbine. A Savonius water turbine was designed by Golecha et al. [12] which has an efficiency of 21%. Savonius turbines have very low manufacturing and maintenance costs compared HAMCT. Still Savonius turbines are not widely used because their efficiency is lower (typically one-half) compared to that of HAMCT. However, the efficiency of Savonius turbine can be improved by placing a duct or an augmentation channel around it. Placing a duct around hydrokinetic turbine can improve turbine efficiency by 70%, as presented in refs. [13, 14]. Kirke [15] performed studies on a Darries hydrokinetic turbine without diffuser and obtained C_P ranging from 0.1-0.25; placing a diffuser increased the C_P to 0.3-0.45 [15]. The concept of ducted Savonius turbine is a novel one; the detailed design and performance analysis of such turbines has not been published in open literature till date.

Computational Fluid Dynamic (CFD) code ANSYS CFX is widely used to compute turbine performance. Kim et al. [16] conducted numerical experiments to study the performance of bi-directional cross flow turbine for tidal power generation. The researchers found that the coefficient of power can be increased significantly by employing a larger area of the channel. Shives and Crawford [17] numerically studied the efficiency of ducted tidal turbines. McSherry et al. [18], Harrison et al. [19], Lain and Osorio [20] and Consul et al. [21] in their respective studies employed numerical methods to investigate the performance of various turbines for tidal energy extraction. Using numerical method allows for rapid design changes which saves time as well as money which otherwise would have been used in the construction of the device.

The current study involves the study on a Savonius rotor housed inside an augmentation channel for tidal energy extraction from uniform, steady current. For numerical simulation, the commercial CFD code ANSYS-CFX was used which is based on Reynolds Averaged Navier-Stokes Equations (RANSE). The code was validated with experimental results. The area ratio and divergence angle of the duct were optimized to increase the turbine efficiency. The performance of the ducted Savonius turbine was determined using numerical method.

NOMENCLATURE

C_P	power coefficient = $P/(0.5\rho AU^3)$
P	rotor power (T x Ω)
T	rotor torque
TSR	Tip speed ratio
U	free stream velocity
Ω	angular velocity of rotor (rad/s)
α	divergence angle
ρ	sea water density

TURBINE DESIGN

Marine current resource assessment was done at one of the marine current streams in Fiji, the location is called Gun-barrel passage, the coordinates of this location are 18°12'1.78"S and 177°38'58.21"E the average marine current speed at this location is 0.85 m/s and maximum velocity exceeds 2.5 m/s [5]. Gun-barrel passage has a mean depth of 17 m, the marine current flow in this stream can have very high turbulence intensity because the passage is quite narrow.

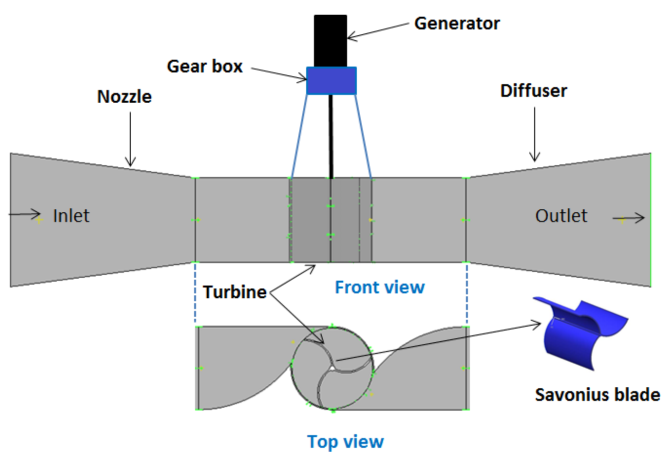

Figure 1. Schematic of Savonius turbine.

A schematic diagram of the ducted Savonius turbine (DST) that is suitable for Gun-barrel passage is shown in Fig. 1. The marine current stream width ranges from 5 to10 m, therefore nozzle (which serves as the augmentation channel) inlet is selected to be 5 m x 5 m. The inlet to throat area ratio and the converging-diverging angles are optimized. A three-bladed Savonius rotor was selected, since three-bladed rotor spins smoothly with minimum vibrations. The rated speed for the turbine is 1.15 m/s, cut-in speed is 0.2 m/s and cut-off speed is 3 m/s. Such turbines can be easily fabricated in PICs because they have simple components that can be easily constructed without the need of advanced manufacturing technologies. The DST can also be installed in low water depths, and in marine streams which have high turbulence intensity. DST does not require frequent maintenance of the turbine blade, because fouling will not have any major effect on its performance; the turbine will produce power as long as the blades are rotating. The generator can be located above the water and maintained regularly. Also screens can be placed at the inlet and outlet to prevent any fish or dirt entering the turbine section.

The schematic of the turbine duct is shown in Fig. 2. The duct was optimized to improve the primary energy conversion efficiency.

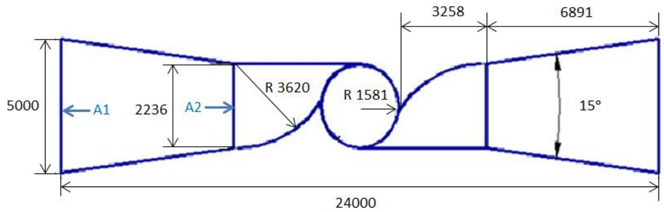

Figure 2. Schematic of duct (augmentation channel).

The area ratio A1/A2 and the converging angle α were optimized. The area ratio was varied from 2 to 10 and α was varied from 5º to 20º; computations were performed at the rated marine current velocity of 1.15 m/s. For each area ratio and α, kinetic energy in the duct was computed using CFX. From the results, the best area ratio obtained was 5 and best α come to 15º. The radii of the guide angle at the turbine inlet and exit were also varied to ensure smooth flow through the duct.

CFD MODELING OF TURBINE

UniGraphics NX 4 CAD package was used for modeling. The inlet has dimensions of 5 m x 5 m and is 24 m long; other dimensions are as shown in Fig. 2. The convergence/divergence angles of the inlet and the outlet sections were 15°. The turbine used in the current study was a 3-bladed Savonius rotor. The parameters of the Savonius rotor are shown in Table 1.

Table 1: Savonius rotor parameters

Parameter	Value
Blade angle	45º
Blade length	158 mm
Outer Diameter	156 mm
Number of Blades	3

ICEM CFD was used for grid generation. The computational domain was discretized with hexahedral grids. The hexahedral grid or user-defined meshing is used to ensure that the obtained results are of the highest quality i.e. high accuracy. Meshing for the turbine is shown in Fig. 3. Only 1/3rd model of the turbine was modeled and meshed. Once the meshing was complete, the 1/3rd was copied using circular array option. This method makes meshing of the complex rotor blade easy.

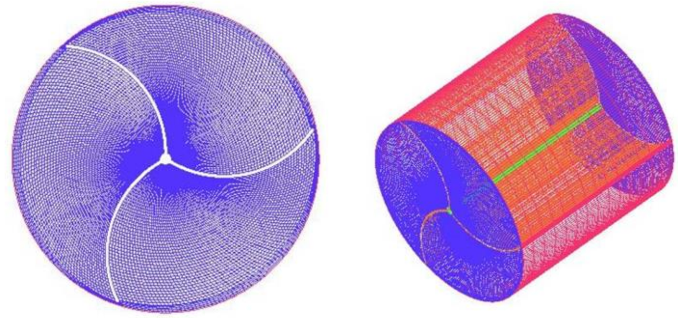
Figure 3. Grid generation for Savonius rotor.

Commercial CFD code ANSYS-CFX was used for the present computations. The sets of equations solved by ANSYS CFX are the unsteady Navier-Stokes equations in their conservation forms. SST turbulence model was used and the simulation type was transient to capture the rotor fluid interaction more accurately. The time discertization of the equations was achieved with the implicit second order Backward Euler scheme.

The computational domain was divided into five sub-domains; inlet, nozzle, diffuser, outlet and turbine as shown in Fig. 4. The inlet, nozzle, diffuser and outlet sub-domains were stationary while the turbine sub-domain was free to rotate about its axis with specified rotational speeds. The entrance of the inlet sub-domain was modeled as an inlet and different velocities were assigned. The exit at the outlet sub-domain was specified with an outlet boundary condition with relative pressure set to 0 Pa. The remaining outside walls of the computational domain were modeled as solid walls where no-slip boundary condition was applied. The no-slip condition ensures that the fluid moving over the solid surface does not have a velocity relative to the surface at the point of contact. Lastly, appropriate interface regions were created. For interface, the mesh connection method was automatic. A scaled model of 1:20 was meshed and same boundary condition was specified to compare the experimental and numerical results. High resolution advection scheme was chosen with high resolution turbulence numerics. For convergence control, maximum coefficient loop was set to 10. The torque was calculated at each time-step and once the torque converged, the simulation was stopped. Great care was exercised while generating the grid. As a check on the accuracy, the results obtained numerically were compared with the experimental data and if the results were way off, the grid was refined. More details on the validation are given in the sections to follow.

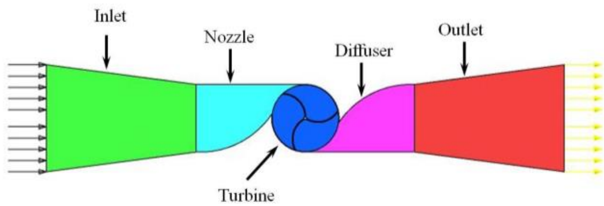

Figure 4. Computational domain.

EXPERIMENTATION

A scaled model of 1:20 was fabricated and tested in a water channel. The dimensions of duct inlet were 0.25 m x 0.25 m x 1.2 m, the diameter of the turbine was 0.156 m. The duct was fabricated using Perspex and blades were made of 2 mm thick aluminum sheet. The fabricated model was tested in a natural water channel 0.8 m x 0.8 m wide and 3 m long. The velocity measurements were taken at 4 locations along the duct without turbine at the free-stream velocity of 0.25 m/s. Then the DST model was carefully placed at the center and allowed to operate at the free-stream velocity of 0.9 m/s. A TEC-2K, 2 kgf-m torque sensor and rpm sensor were connected to the turbine blade shaft and the output was computed on DANCELL digital indicator. The torque was recorded at three rotational speeds of 9, 15 and 21 rpm.

EXPERIMENTS AND CFD VALIDATION

The velocity vectors in the XY plane for the model without turbine are shown in Fig. 5. For this simulation (without the turbine), the inlet velocity was 0.2 m/s. The velocity increased as the water flowed from the inlet to the nozzle as expected. At region A the increase in velocity is significant and water mostly passes through the turbine housing on the upper surface which in turn leaves a re-circulating flow in region marked as B. A large vortex flow is also observed in the diffuser in region C.

The velocities recorded along the length of the tidal turbine model namely at points 1 to 4 are shown in Fig. 6. For the simulation with the turbine model, the inlet velocity was 0.6 m/s. Point 1 is a point near the inlet and point 2 is the point close to the nozzle entrance. The velocity increases from point 1 to point 2. This indicates converging flow and as the cross sectional area reduces, the velocity increases. This trend is observed for both the cases of with and without the turbine. On the other hand, point 3 is near the diffuser exit and point 4 is at the outlet. Looking at the case without the turbine, the velocity reduces as the water flows through the diffuser into the outlet. The velocity remains fairly constant. For the case with turbine, the velocity at point 3 is slightly higher because of the formation of unsteady vortex.

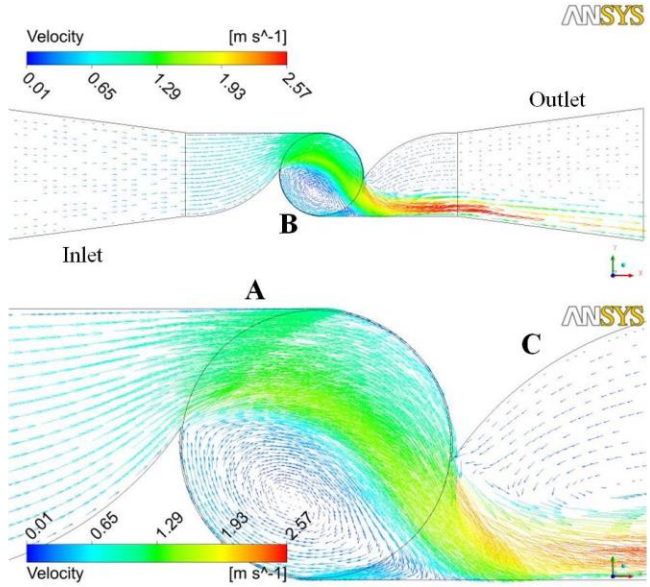

Figure 5. Velocity vector in XY plane for model without turbine.

The other reason for this is – due to the presence of the turbine, the water flowing through the turbine and into the diffuser is forced further towards the bottom of the diffuser wall which shows some flow acceleration in this region. For the case without the turbine, there is a very good agreement between experimental and CFD results for all the four points; however, for the case with the turbine, the velocities at point 3 and point 4 are slightly over-predicted by CFD, this is mainly due to velocity fluctuations at points 3 and 4.

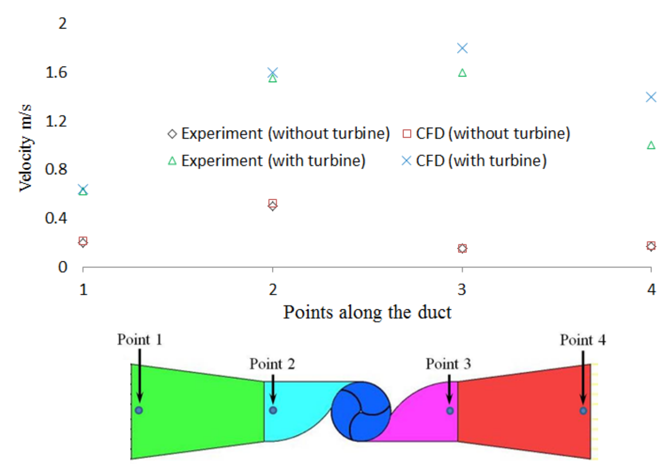

Figure 6. Velocities at points 1 to 4 for the cases with and without the turbine.

Figure 7 shows a comparison of the power coefficient obtained from experimentation and numerical work at very low rotational speeds. The torque obtained by CFD is over-predicted by 5%; otherwise experimental and CFD results are in good agreement. Therefore CFD (ANSYS-CFX) was used to predict the performance of the full-size turbine.

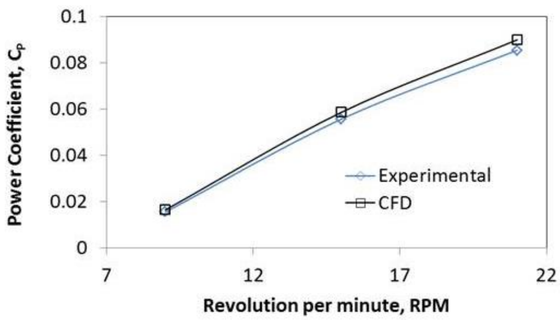

Figure 7. Comparison of power coefficient obtained experimentally and numerically at very low rotational speeds.

PERFORMANCE ANALYSIS OF FULL-SIZE TURBINE

The turbine performance and rotor power were computed numerically using CFX. Turbine simulations were carried out at different TSR and at different inlet velocities; for each case, average torque over the blade was computed. The performance of DST is shown in Fig. 8. The turbine has maximum efficiency of 53% at the TSR of 3.5 at the rated speed of 1.15 m/s, 52% at the free-stream velocity of 0.5 m/s, and 50% at 2.15 m/s. The turbine efficiency is significantly higher for TSR between 3-4. As shown in Fig. 8, the efficiency increases, reaches its maximum and then decreases significantly. The peak in efficiency basically indicates that the interaction between the turbine and the flow is maximized at this optimum TSR. At this TSR, maximum energy is extracted hence higher turbine power and efficiency.

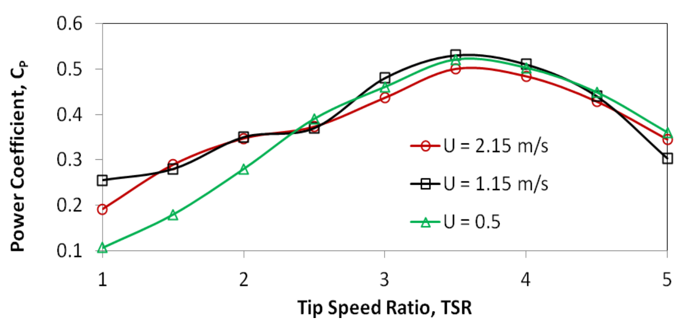

Figure 8. Turbine performance at different TSR and free stream velocity

The maximum power of the rotor is 10 kW at the rated speed of 1.15 m/s, 0.83 kW at 0.5 m/s and 63.4 kW at 2.15 m/s. The turbine starts producing power at low speeds; therefore this turbine can also be used at locations where marine current velocity is low. An increase in the turbine efficiency is recorded when a duct is placed around the Savonius turbine; the duct significantly reduces the pressure at the back of the turbine hence improving the turbine performance.

CONCLUSIONS

A ducted Savonius turbine which can operate at lower velocities and low water depths was successfully designed and tested for performance. The performance of the DST is improved due to the duct. Current studies show DST can achieve maximum efficiency up to 53% at TSR of 3.5, and corresponding maximum power of 63.4 kW at 2.15 m/s. DST can be easily manufactured with low cost and the maintenance cost is also low. Such turbine is appropriate for PICs like Fiji to generate electricity and feed into the grid.

ACKNOWLEDGMENTS

The authors would like to acknowledge the help rendered by Mr. Shiu Prasad, Senior Technician at the University of the South Pacific and Mr. Sandeep Kumar, Mr. Teebwatia Taakau, and Mr Josaia Koroilavesau, final year students, 2012, for helping in the fabrication of the turbine model and during experimentation.

REFERENCES

[1] "Fiji Electricity authority; annual report 2011, 2011" *Fiji electricity authority*, available: http://www.fea.com.fj/pages.cfm/downloads/annualreports.html.

[2] Batten, W.M.J., Bahaj, A.S., Molland, A.F. and Chaplin, J.R., 2006, "The prediction of Hydrodynamic performance of marine current turbines," *Renewable Energy*, Vol. 33, pp. 1085-1096.

[3] Sale, D., Jolman, J. and Musial, W., 2009, "Hydrodynamic optimization method and design code for stall- regulated hydrokinetic turbine rotor," *National Renewable energy laboratory*.

[4] Nicholls-Lee, R.F., Turnock, S.R. and Boyd S.W., 2008, "Simulation based optimisation of marine current turbine blades," *7th International Conference on Computer and IT Applications in the Maritime Industries*, Liège, Belgium.

[5] Goundar. J.N. and Ahmed, M.R,. 2012, "Marine Current Energy Resource Assessment and Design of a Marine Current Turbine for Fiji," *Asia-Pacific Forum on Renewable Energy 2012*, jeju, Korea.

[6] Maganga, F. and Germain, G., 2010 "Experimental characterisation of flow effects on marine current turbine behaviour and on its wake properties," *Let Renewable Power Generation*, Vol 4, pp. 498-509.

[7] Zhu, T. and Rempfer, D., 2012 "Numerical study of detailed flow field and performance of Savonius wind turbines," *Renewable Energy*, vol. 51, pp. 373-381.

[8] Shigettomi, A., Murai, Y., Tasushi, Y. and Takeda, Y., 2011, "Interactive flow field around two Savonius turbines," *Renewable Energy*, vol.36, pp. 536-545.

[9] Saha, U. K, and Rajkumar, M. J., 2006, "On the performance analysis of Savonius rotor with twisted blades," *Renewable Energy*, vol. 31, pp. 1776-1788.

[10] Kamoji, M. A., Kedare, S. B. and Prabhu, S. V., 2009 "Experimental investigation on single stage modified Savonius rotor," *Applied Energy*, vol. 86, pp. 1064-1073.

[11] Mohamed M. H., Janiga, G., Pap, E. and Thevenin, D., 2010 "Optimisation of Savonius turbine using an obstacle shielding the returning blade," *Renewable Energy*, vol. 35, pp. 2618-2626

[12] Golecha, K., Eldho, T.I. and Prabhu, S. V., 2011, "Influence of the deflector plate on the performance of modified Savonius water turbine," *Applied Energy*, vol. 88 pp. 3207-3217

[13] Coiro, D.P., 2012, "Development of Innovative Tidal Current Energy Converters: From Research to Deployment," *Asia-Pacific Forum on Renewable Energy 2012*, jeju, Korea.

[14] Khunthongjan, P. and Janyalertadun, A., 2012, "A study of diffuser angle effect on ducted water current turbine performance using CFD," *Songklanakarin Journal of Science and Technology*, vol. 34 pp. 61-67.

[15] Kirke, B. K., 2011, "Tests on ducted and bare helical and straight blade Darrieus hydrokinetic turbines," *Renewable Energy*, vol. 36, pp. 3013-3022.

[16] Kim, K. P., Ahmed, M. R. and Lee, Y. H., 2012, "Efficiency Improvement of a Tidal Current Turbine Utilizing a Larger Area of Channel," *Renewable Energy*, Vol. 48, pp. 557-564, 2012.

[17] Shives, M. and Crawford, C., 2010, "Overall Efficiency of Ducted Tidal Current Turbines," *Proceedings of the OCEANS 2010 MTS/IEEE SEATTLE Conference & Exhibition*, seattle, *WA*.

[18] McSherry, R., Grimwade, J., Jones, I., Mathias, S., Wells, A. and Mateus, A., 2011, "3D CFD Modelling of Tidal Turbine Performance with Validation Against Laboratory Experiments," *Proceedings of 9th European Wave and Tidal Energy Conference*, University of Southampton, United Kingdom

[19] Harrison, M. E., Batten, W. M. J., Myers, L. E. and Bahaj, A. S., 2009, "A Comparison between CFD and Experiments for Predicting the Far Wake of Horizontal Axis Tidal Turbines," *Proceeding of 8th European Wave and Tidal Energy Conference*, Uppsala, Sweden.

[20] Lain, S. and Osorio, C., 2010 "Simulation and Evaluation of a Straight-Bladed Darrieus –Type Cross Flow Marine Turbine," *J. Scientific & Industrial Research*, Vol. 69, pp. 906-912.

[21] Consul, C. A., Willden, R. H. H., Ferrer, E. and McCulloch, M. D., 2009, "Influence of Solidity on the Performance of a Cross Flow Turbine," *Proceedings of 8th European Wave and Tidal Energy Conference*, Uppsala, Sweden.