*International*
*Journal*
*Of Advanced Research in Engineering & Management (IJAREM)*
*ISSN: 2456-2033 || PP. 06-12*

# A Review On Numerical Analysis and Parametric Study of Cross Flow Turbine

Anuj Agarwal[a] , Dr.Vilas Warudkar[b]
[a]*Department of mechanical engineering ,M.tech scholar, MANIT Bhopal,462003,Madhya Pradesh.*
[b] *Department of mechanical engineering ,Asst.Professor ,MANIT,Bhopal, 462003,Madhya Pradesh.*

**ABSTRACT:** There is a growing interest in utilizing tidal current energy for power generation which has led to extensive research on this source of renewable energy. The work presented in this project aims to study the tidal current energy extraction using a cross-flow turbine. The bi-directional flow of tidal currents is used to drive a uni-directional cross-flow turbine. Thus in present study cross-flow turbine studied is placed in an augmentation channel, having a nozzle and a diffuser. The performance of the device is studied numerically using the commercial ANSYS Fluent code while the model is being built using SOLIDWORKS. The internal flow characteristics of the turbine are studied for various cases. Results of the numerical analysis are presented in terms of pressure contours, streamlines and velocity vectors. A good agreement was observed between literature and present study .A comparative study has been performed by changing the turbine inlet profile from parabolic to invert parabolic in order to study the impact on pressure and velocity change. Moreover a parametric study of variations in TSR ratio and radius of blades is done for their effects on Power coefficient and Turbine Torque. The tip speed ratio was varied from 0.85- 1.4.The analysis from above study can be used for enhancement of performance of cross flow turbine.

**Keywords:** Cross flow turbine, SOLIDWORKS, ANSYS, Fluent, Parametric Study, Power coefficient,

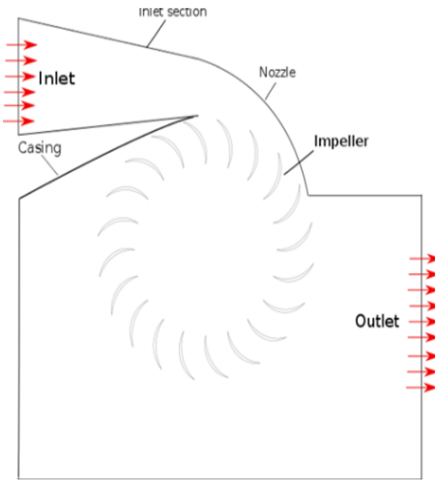
## 1. INTRODUCTION

The urgent need to establish a clean, safe and affordable energy supply has placed an increased emphasis on the exploitation of new renewable energysources. The ocean offers immense potential for clean energy extraction and besides wave power and tidal barrage technologies, tidal stream turbines have been identified as prospective marine energy converters. This presents numerical investigations of the hydrodynamics of generic marine cross-flowturbines. Evidence suggests that global energy generation, currently dominated by fossil fuels, is one of the prime sources for causing or at least accelerating climate change. A strong correlation between the earth’s average temperature and the global concentration of carbon dioxide has been demonstrated. Another key factor is energy security that is how to reliably match demand and supply in years to come. These aspects underline why the renewable energy sector has received renewed attention over the last few decades; renewable energy is expected to play a pivotal role with regard to solving the challenges of climate change and establishing a safe and affordable energy supply. The UK has manifested its support of the renewable energy sector by committing to ambitious goals, such as 20% of the country’s electricity come from renewable sources by 2020, and reducing carbon dioxide emissions based on 1990 levels by 60% by 2050, as outlined in DTI (2003). While the electricity generated from renewable sources in the India has increased from 10 KWh/yr in 2000 to 25 KWh/yr in 2010, it is still only about 7% of the India’s electricity generation that came from renewable sources in 2010. In order to achieve its goals the India is required to significantly increase the contribution from renewable sources to the energy supply over the next 10 years. So far, the focus of investors, developers as well as the media has primarily laid on biofuels, wind and solar energy. More recently, however, the vast untapped energy source of the ocean has also caught the interest of the researchers. It was seen that previous researches were done on bi-directional cross flow turbine and augmentation channel strongly influenced the flow and the turbine performance. In addition, the flow resistance by the augmentation channel and the blades forced the flow to divert away from the augmentation channel, reducing the flow velocity and turbine efficiency. Most of the literatures focused on experimental study which were restricted to many constraints like high cost and constant parameters turbine model. Not much literature were focused on 3-D simulation of cross flow turbine and they ignored effects of various parameters such as TSR, blade radius and channel shape on the performance of turbine. Also parametric study for various parameters is missing in many literatures. In this study, performing of CFD analysis of turbine using ANSYS Fluent and studying various parameters like TSR ratio, radius of blades and turbine channel profile and their effects on Power coefficient and Turbine Torque is done.

## 2. CROSS FLOW TURBINE ANALYSIS AND STUDY

### A. CROSS FLOW TURBINE

A cross–flow turbine in its simplest form consists of a runner and a nozzle.



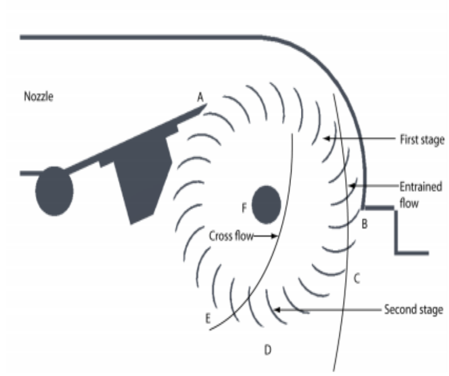



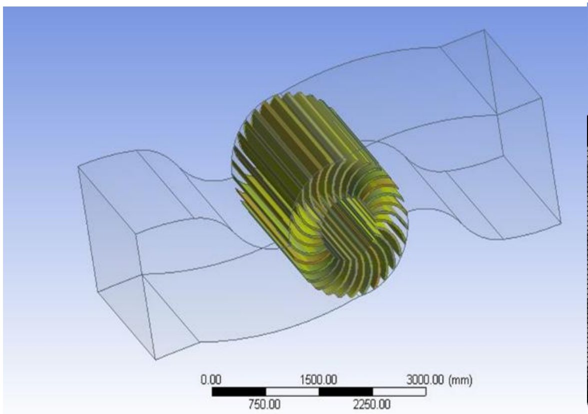

Figure 1: Schematic of Cross flow turbineFigure 2: Working of cross flow Turbine

The peak efficiency of a cross-flow turbine is somewhat less than a Kaplan, Francis or Pelton turbine. However, the cross-flow turbine has a flat efficiency curve under varying load. With a split runner and turbine chamber, the turbine maintains its efficiency while the flow and load vary from 1/6 to the maximum.

Since it has a low price, and good regulation, cross-flow turbines are mostly used in mini and micro hydropower units of less than two thousand kW and with heads less than 200 m.

### B. NUMERICAL ANALYSIS

Here it is chosen to use the CFD software ANSYS FLUENT 14.5.0, which solves the Reynolds-Averaged Navier-Stokes (RANS) equations, using a finite volume approach. For this study, ANSYS FLUENT is used as a three-dimensional, pressure-based, segregated, implicit, incompressible flow solver.



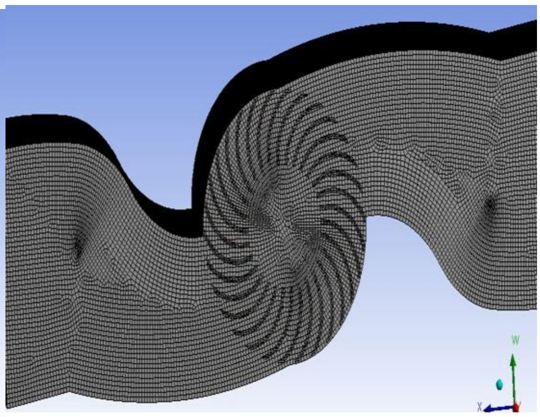



Figure 3: Ductless Model of Cross flow turbineFigure 4: Meshed model of Cross flow turbine

The CFD analysis of cross flow turbine was performed using ANSYS Fluent and thus obtaining various plots of Velocity, pressure and streamline using ANSYS CFD post.

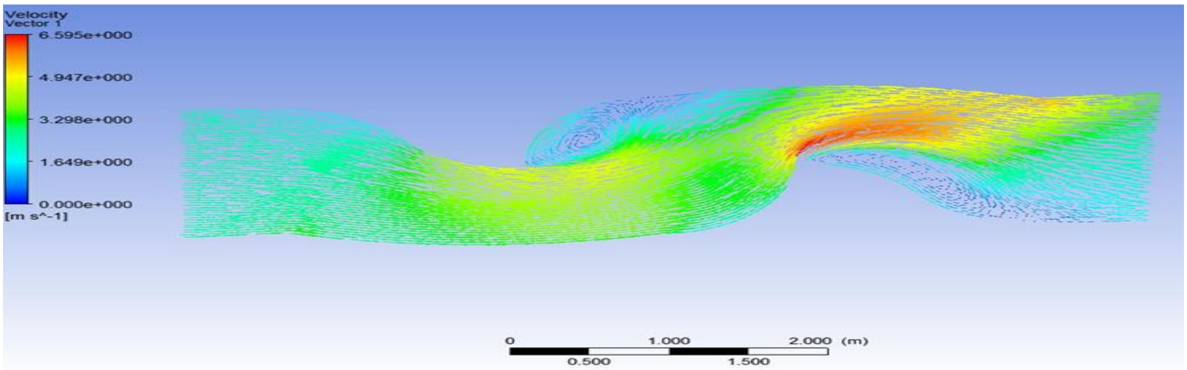



Figure 5: Velocity plot

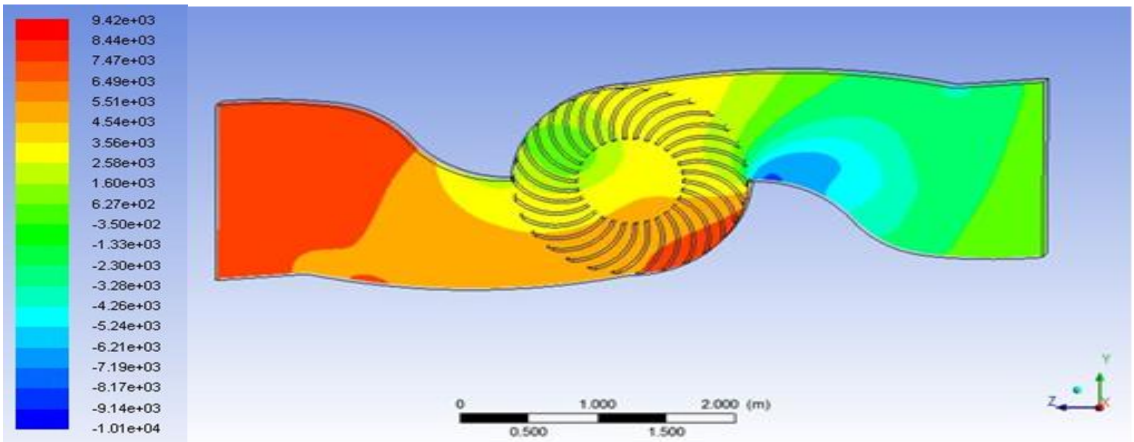



Figure 6: Pressure plot

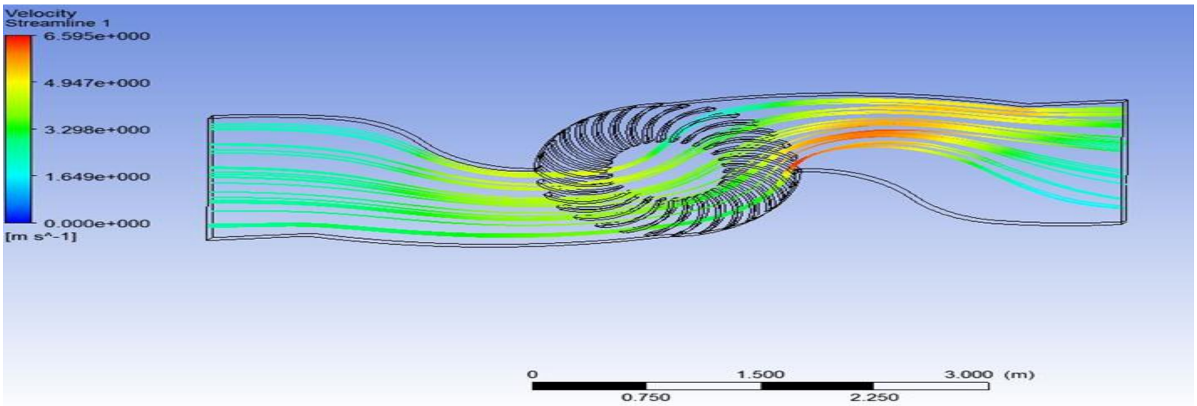



Figure 7: Streamline plot

It was observed that maximum velocity **was 6.59m/s** and maximum pressure **9420Pa**from fig. 5 and fig. 6 respectively.

## C. PARAMETRIC STUDY

The internal diameter of cross flow turbine is being varied and a parametric study is being performed to study its effects on turbine performance**.**

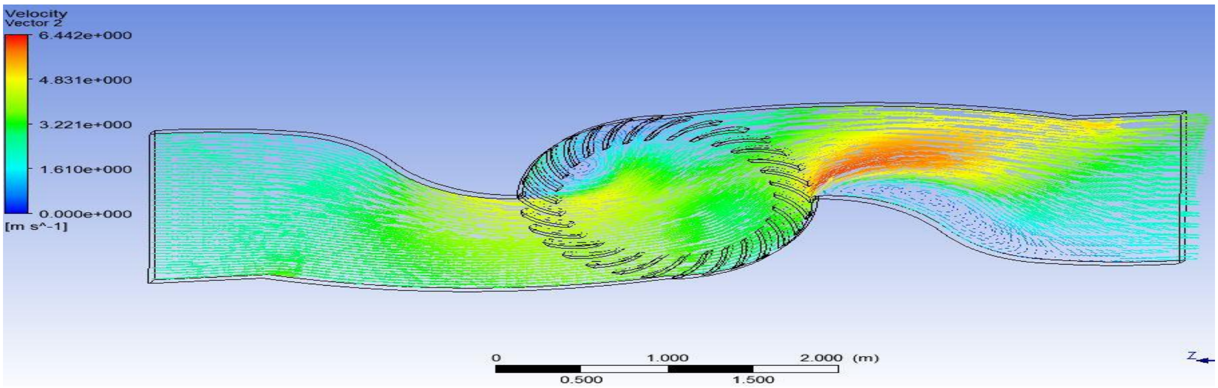


Figure 8: Velocity vector for increased internal runner diameter and decreased blade size

**D. COMPARITIVE STUDY**

For comparative study the flow to turbine is being diverted through parabolic channel as compared to inverted parabolic in study. In order to study the impact on pressure and velocity change.

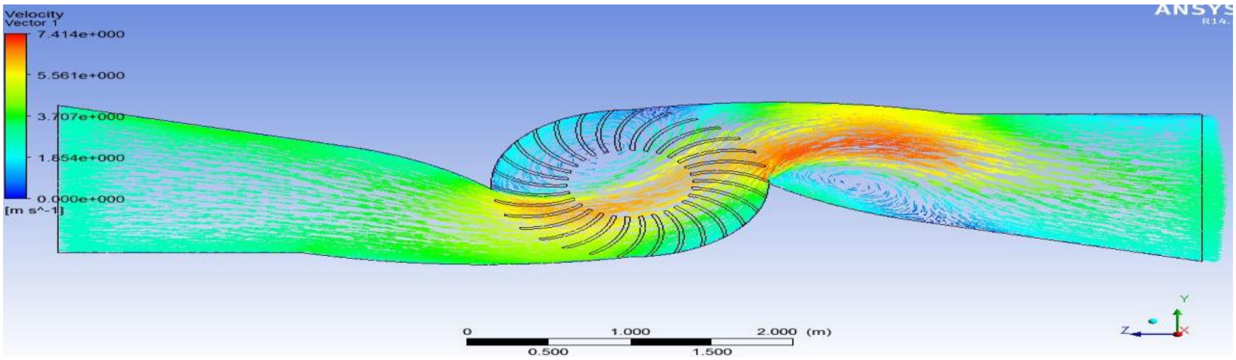


Figure 9: Velocity Contours for inverted parabolic channel flow

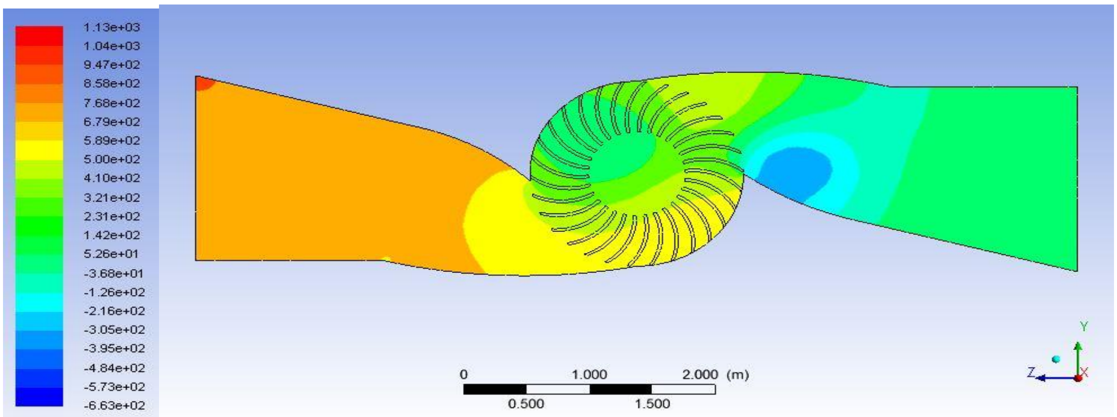


Figure 10: Pressure Contour for inverted parabolic channel

From the above pressure plot it can be observed that contours of pressure have become smooth as compared to previous study. But it can also be observed that the maximum pressure reaches to **1130 Pa** as compared to **9420 Pa** as was observed previously. Moreover maximum velocity at exit of runner blades reaches to **7.41m/sec**.

## 3. HIGHLIGHTS OF WORK DONE ON CROSS FLOW TURBINE

Highlightedpoints of the research work done in the field of cross flow turbine are listed below in front oftheir research topic.

### A. Experimental and analytical study of helical cross-flow turbines for a tidal micropower generation system.

This study investigates the feasibility of a micro-scale tidal hydrokinetic generator to power autonomousoceanographic instrumentation, with emphasis on turbine design and performance. This type of "micropower"system is intended to provide continuous power on the order of 20 Watts. A steady-state modelpredicts system energy storage and power output in a mixed, mainly semidiurnal tidal regime with peak currents of 1.5 m/s. Parameters impacting helicalturbine design include radius, blade profile and pitch, aspect ratio, helical pitch, number of blades, solidity ratio,blade wrap ratio, strut design, and shaft diameter. Maximum turbine efficiency increased with increased flume velocity. A free-vortex model was modified to simulate the helical turbine performance.Model results were compared to experimental data for various strut design and inflow velocities, andperformance was extrapolated to higher flume velocities and a full-scale turbine (0.7 m2 relative to 0.04 m2 influme tests). The model predicts experimental trends correctly but deviates from experimental values for someconditions, indicating the need for further study of secondary effects for a high chord-to-radius ratio turbine.

### B. Experimental study of cross-flow turbine.

Here in this research paper there is study based on experimental performance of cross flow turbine by varying the number of blades, the runner diameter, and the nozzle entry arc under flow/head variations. The results, as presented in this paper, show that the maximum efficiency of the CFT at any flow/head combination increases as the nozzleentry arc increases or the aspect ratio of the runner decreases. Equations have been suggested to obtain the optimum number of blades for maximum efficiency of the turbine and to find the specific speed of the CFT as a function of unit discharge and nozzle entry arc.

### C. Power outputs of two stages of cross-flow turbine.

Here in this review there is study of the contribution of the two stages of powergeneration to the shaft power in a cross-flow turbine. A special model of the turbine is built to allow insertion of flow diverter inside the runner. A flow diverter is alsodesigned to intercept the flow that passes through the first stage of the turbine. Tests are conducted with and without the diverter to determine the contribution of the two stages of the turbine to shaft output. Measurements are taken to ensure that presence of the flow diverter does not cause back pressure inside the nozzle. Contribution of the first stage to output is determined from tests with the diverter installed. Total shaft power is determined by removing the diverter. The balance between the two is the contribution of the second stage. The turbine's second stage contributes more significantly to the shaft output than reported in the analytic literature.

### D. Investigation of the performance of a cross-flow turbine.

In this paper an experimental investigation was conducted to study the effects of some geometric parameters of runners and nozzles(e.g., diameter ratio and throat width ratio) on the efficiency in the cross-flow turbines, by varying of ratio ofinner-to-outer diameters of runners and gate openings of two different turbine nozzles under different heads. In this study, four different types of runners (170 mm outer diameter, 114 mm width) were designed and manufactured to investigate the effects of the ratio of inner-to-outer diameters of runners on the turbine efficiency. Each runner had 28 blades and the ratios of inner-to-outer diameters of runners were 0)75, 0)67, 0)58 and 0)54, respectively. The runners weredenoted with the numbers 1, 2, 3 and 4, and nozzles *A* and *B*. The performance parameters namely output power, efficiency, runawayspeed, reduced speed and power for different nozzle/runner combinations were investigated by changing head range from 8 to 30 m, the nozzle *A*-runner combinations (A—1, 2, 3, 4) and from 4 to 17 m, the nozzle *B*-runner combination (B—2) at different gate openings. The results of the present study clearly indicated that there was a negligible difference (e.g., 3% intotal between 0)54 and 0)75 diameter ratio) in the efficiency of turbine for different diameter ratios and heads, and that the highest efficiency was obtained as 72% for A—2.

**E. Study of the nozzle flow in a cross-flow turbine.**

In this research paper there is an investigation of the flow inside the nozzle of a Cross Flow turbine, which is a hydraulic turbine where the rectangular water jet issuingfrom the nozzle crosses the rotor blades twice. Part of the investigation consisted in the experimental measurement of the static pressure distribution on the inside walls of two different nozzle configurations, both with the nozzle mounted alone and in the presence of rotor. The analysis of the results obtained in this way gives an indication of the influence of the turbine non-dimensional volume flow rate on the flow inside the nozzle and the way it affects the reaction degree of the machine and its efficiencylevel.

**F. A numerical method of free jet from a cross-flow turbine nozzle.**

The nozzle of cross flow turbine has to give a certain circumferential velocity and an optimum angle to the flow at the nozzle exit (runner exit). Therefore, the nozzle shape has an important influence upon the turbine performance. Its shape is asymmetric and complicated, and the exit flow from it has free boundaries. The flow from such a nozzle has not been analyzed enough until now because of complexity of its flow. In this paper, the flow from a nozzle with arbitrary asymmetric curved surfaces is calculated numerically by Schwarz-Christoffel method. In order to estimate the accuracy of this method, the calculation results of the flow of cross flow turbine nozzle are compared with experimental ones. Also the effects of nozzle shape on the exit flow are investigated.

**G. Performance of horizontal axis tidal current turbine by blade configuration.**

In this study they investigated the blade of the rotor is one of the essential componentswhich can convert tidal current energy into rotational energy to generate electricity. The variable blade properties determine the performance, efficiency and stability of the turbine system. This paper presents the design procedure for a 300 kW tidal current turbine blade. The HAT turbine model was designed based on the wind mill turbine design principles together with known turbine theories. To verify the compatibility of the turbine design method and to analyze the properties of design factors, the 3D CFD model was applied with the ANSYS CFX program. The characteristics and theperformances of the blades can be applied in the design of 300 KW and larger capacityTCP rotors.

**H. CFD study of a ducted cross flow turbine concept for high efficiency tidal current energy extraction.**

Here it is studied that the tidal current energy extraction using a cross-flow turbine.This study focused on the influence on the turbine performance according to the variation of the duct size turbine. The cross-flow turbine studied is placed in an augmentation channel, having a nozzle and a diffuser. The performance of the device is studied numerically using the commercial ANSYS-CFX code. Results of the numerical analysis are presented in terms of pressure contours, streak lines and velocity. It was seen the optimum tip speed ratio for the 2.25A duct size case was at TSR of 0.99.In addition, mixing effects between the faster flow at the outside the casing and the outlet flow may have increased flow speed of the outlet flow due to thegap and may have improved performance.

## 4. CONCLUSIONS

In this broad review all important aspects related to cross flow turbine, its numerical analysis and different studies such as parametric and comparative is mentioned. Someplots such as velocity, pressure etc. are discussedand the main aim of the work is exploration in field of the tidal current energy extraction using a cross-flow turbine. A 3D model and its analysis and study will develop better understanding in this field so that more exploration could be performed using tidal energy.

## REFERENCES

[1]. Garrett, C. and Cummins, P., 2007. The efficiency of a turbine in a tidal channel. Journal of fluid mechanics, 588, pp.243-251.
[2]. Khalid, S.S., Liang, Z. and Shah, N., 2012. Harnessing tidal energy using vertical axis tidal turbine. Research Journal of Applied Sciences, Engineering and Technology, 5(1), pp.239-252.
[3]. Niblick, A.L., 2012. Experimental and analytical study of helical cross-flow turbines for a tidal

micropower generation system (Doctoral dissertation, University of Washington).
[4]. Khosrowpanah, S., Fiuzat, A.A. and Albertson, M.L., 1988. Experimental study of cross-flow turbine. Journal of Hydraulic Engineering, 114(3), pp.299-314.
[5]. Fiuzat, A.A. and Akerkar, B.P., 1991. Power outputs of two stages of cross-flow turbine. Journal of energy engineering, 117(2), pp.57-70.
[6]. Olgun, H., 1998. Investigation of the performance of a cross-flow turbine.International Journal of Energy Research, 22(11), pp.953-964.
[7]. Pereira, N.C. and Borges, J.E., 1996. Study of the nozzle flow in a cross-flow turbine. International journal of mechanical sciences, 38(3), pp.283-302.
[8]. FUKUTOMI, J., NAKASE, Y. and Watanabe, T., 1985. A numerical method of free jet from a cross-flow turbine nozzle. Bulletin of JSME, 28(241), pp.1436-1440.
[9]. Kelso, R.M., Lim, T.T. and Perry, A.E., 1996. An experimental study of round jets in cross-flow. Journal of fluid mechanics, 306, pp.111-144.
[10]. Hee Jo, C., young Yim, J., hee Lee, K. and ho Rho, Y., 2012. Performance of horizontal axis tidal current turbine by blade configuration. Renewable Energy, 42, pp.195-206.
[11]. Lain, S. and Osorio, C., 2010. Simulation and evaluation of a straight-bladed Darrieus-type cross flow marine turbine. Journal of Scientific & Industrial Research, 69(12), pp.906-912.
[12]. Schlüter, J.U. and Schönfeld, T., 2000. LES of jets in cross flow and its application to a gas turbine burner. Flow, Turbulence and Combustion, 65(2), pp.177-203.
[13]. Kim, I.C., Wata, J., Ahmed, M.R. and Lee, Y.H., 2012. CFD study of a ducted cross flow turbine concept for high efficiency tidal current energy extraction. In Proceedings of Asian Wave and Tidal Energy Conference 2012(pp. 400-405).
[14]. Polagye, B., Cavagnaro, R., Niblick, A., Hall, T., Thomson, J. and Aliseda, A., 2013. Cross-flow turbine performance and wake characterization. InProceedings of the 1st Marine Energy Technology Symposium, Washington, DC.