Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 1, Number 3; October, 2014 pp. 55-59
© Krishi Sanskriti Publications
http://www.krishisanskriti.org/jbaer.html

Design and Analysis of a Cross Flow Hydrokinetic Turbine Using Computational Fluid Dynamics

Himanshu Joshi[1], Arpit Dwivedi[2], Anish Anand[3], Pravin P. Patil[4]
[1,2,3]B.Tech (M.E), Graphic Era University, Dehradun
[4]Graphic Era University, Dehradun

***Abstract:** The revolution in cross flow turbine industry came when the straight blades of the Darrieus turbine were modified into helical shape by Alexander M. Gorlov. There have been several research projects dealing with the design and analysis for tidal applications. This paper deals with the design and analysis of a cross flow hydrokinetic turbine (CFHT) with helical blades. Static analysis with optimum blade velocity and constant pressure conditions was performed for the blade with fixed pitch by using Computational Fluid Dynamics (CFD) in Fluent v14.5. Catia v5 was used to carry out 3D modeling of the turbine. The hydrofoil shape of NACA 0018 was created by the airfoil coordinate database. Two different turbulence models Spalart-Allmaras (One-Equation model) and sst-k□ (Two –Equation model) were employed to compute and compare the results. Pressure profiles, drag and lift coefficients are calculated under a steady flow of 1.5 m/s.*

1. INTRODUCTION

Generation of electricity in a cheap and efficient manner is the primary goal of every country. Each power generating company tries to develop the best suitable method to generate electricity and renewables have become the primary source of power generation with the rapid consumption of conventional fuel. Hydropower, large and small, remains by far the most important of the "renewables" for electrical power production worldwide, providing 19% of the planet's electricity [1]. The flow potential of water currents in rivers, oceans, estuaries is thus studied as an immaculate and environmentally amiable source. India has a hydro- potential of 250000 MW yet only 14.75 % of it is in use, i.e. it has an installed capacity of 36848 MW [2]. There is a large demand of clean and economically viable form of energy. CFHT (Gorlov turbine) provides that alternative. It rotates at twice the velocity of water current flow and in the same direction independent of water flow direction [3]. The turbine is modular in design i.e. it can be assembled vertically, horizontally or in any other cross flow combination using a common shaft and generator for an array of multiple turbines. This helps in reducing the construction, expansion and maintenance costs for any such power generating unit.

The NACA 0018 hydrofoil used as the blade profile is symmetric and has an 18% width-to-thickness ratio. This symmetrical hydrofoil shape helps in balancing the forces generated during the rotation of the turbine, as the direction of forces changes after 180º of rotation. Static analysis refers to the simple analysis in the presence of flow without any specific rotation of the turbine. Before manufacturing the turbine it is essential to study the flow parameters such as velocity, pressure, shear stress distribution, turbulent kinetic energy. Present work highlights the design and analysis for a model of helical blade cross flow turbine (CFHT). The analysis was done for a scaled model of height 230mm, 4 bladed turbine with 58.5º inclination angle. Pressure contours, drag and lift coefficient are calculated under a steady flow of 1.5 m/s using CFD solver fluent 14.5.

Hydrokinetic turbines are in early stage of development. Much of the theory, concepts associated with Hydrokinetic turbine are derived from the wind industry. The technology is where the wind energy industry was approximately three decades ago, with many developments to come [4]. Taylor Jessica Hall [5] studied the numerical simulation of a marine cross flow hydrokinetic turbine; she studied the static and dynamic torque analysis on a model of turbine by varying tip speed ratios. Adam A. Niblick [6] studied the experimental and analytical study of cross-flow turbines for a tidal micropower generation system. J. Zanette, D. Imbault [7] studied a design methodology for cross-flow turbines. Claudio A Consul [8] studied the influence of turbine solidity on turbine performance and the influence of blockage and free surface deformation on the hydrodynamics of energy extraction. Samantha Jane Adamski [9] studied the Numerical Modeling of the Effects of a Free Surface on the Operating Characteristics of Marine Hydrokinetic Turbines.

CFD simulations do not need any external data (experimental lift & drag) and can include separation from foils and drag induces vortices from turbine's shaft.CFD modeling is a powerful tool for complex geometries. However, CFD simulations for tidal turbines still suffer from high computational cost and time, thus it is very important to analyze the problem first. We have solved till 150 iterations for each turbulent model independently. When a blade rotates, its angle of attack (α) (angle between local relative velocity and chord) changes leading to variable hydrodynamic forces.

Spalart-Allmaras model is a one equation model which solves a transport equation for a viscosity like variable ν. This may be referred to as the Spalart-Allmaras variable. The SST k-ω turbulence model [10] is a two-equation eddy-viscosity model which has become very popular. The shear stress transport (SST) formulation combines best of two worlds. The use of a k-ω formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer. The SST formulation also switches to a k-ε behavior in the free-stream and thereby avoids the common k-ω problem that the model is too sensitive to the inlet free-stream turbulence properties. Figure 1 shows the distribution of forces and pressure on the hydrofoil geometry at different angles.

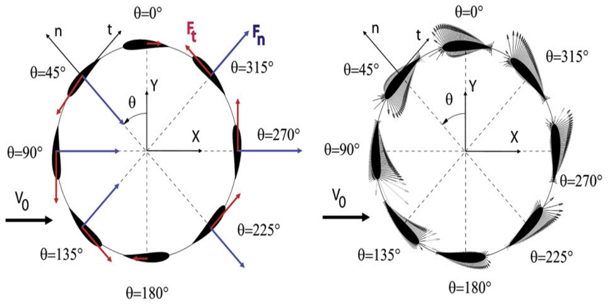

Figure. 1 Schematics of forces and pressure vectors on an airfoil in a cross flow turbine. [7]

2. CAD MODEL AND MESHING DETAILS OF THE TURBINE

The helical profile of the turbine blade was created using the data points obtained from UIUC Airfoil Coordinate database [11]. A reference model, a rectangular channel was created and the continuum used is water. The CAD model is shown in figure 2 and the details of the geometry are mentioned in Table 1.

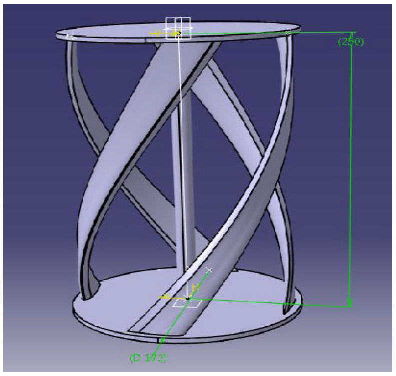

Figure. 2 CAD model

In figure ‘3’ turbine is placed inside the channel which was developed in ANSYS 14.5 workspace .It is used as a duct through which water at normal atmospheric conditions will flow, thus water act as a continuum.

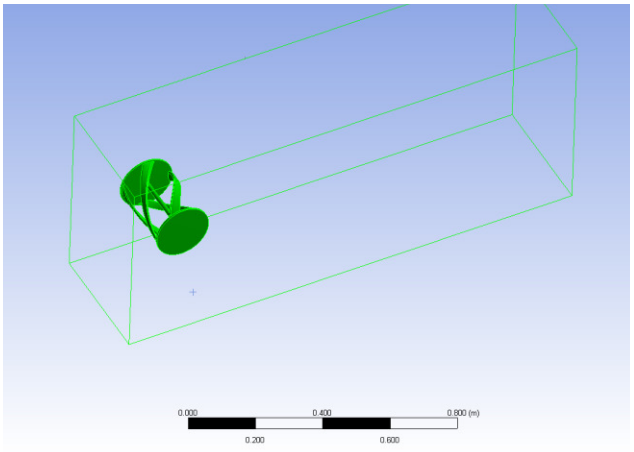

Figure. 3 Turbine inside the channel

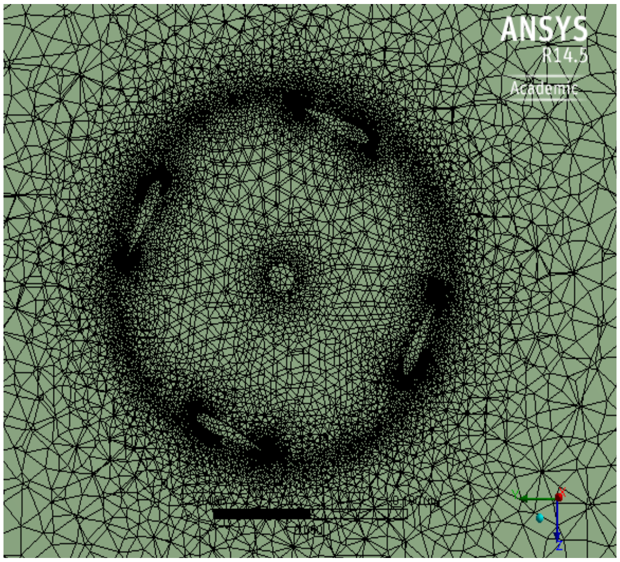

Figure. 4 Cross-section of the curved mesh

The CAD model of the turbine consists of four helical blades, two circular plates each 4 mm thick and a central shaft that would be coupled with generator. The meshing was done with fine relevance center as shown in fig. 2. The meshed model consists of 3, 77, 697 nodes and 2, 28, 341 elements.

The sweep in the helical blade geometry is given by the inclination angle (□) which is a measure of inclination of the blade with respect to the horizontal plane. An inclination of 90º indicates a straight blade as found in the Darrieus type turbine. In general, the closer the inclination angle is to 90º the better is the efficiency of the turbine [12].

Table 1. Blade and turbine parameters

S. No	Turbine Parameter	Magnitude
1	Blade profile	NACA0018
2	Number of blades, N	4
3	Chord length, c	0.04m
4	Diameter	0.172m
5	Height of turbine	0.230m
6	Inner shaft Diameter	0.007129m
7	Inclination angle	58.49º
8	Aspect ratio (A.R)	1.3372
9	Solidity Ratio (σ) for 4 blade	0.29625
10	Plate thickness	0.004m
11	Moment Centre	(0.194, 214.0)

3. MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

There are a wide variety of materials by which turbine blades are manufactured. We have considered 5086 marine grade aluminum, primarily alloyed with magnesium. It has good corrosion resistance properties and it has a density of 2, 660 kg/m3 slightly less dense than aluminum.

Boundary conditions employed in computations consists of a constant velocity inlet of 1.5 m/s on left side, a constant pressure outlet of 104268 Pa on right, wall conditions include no slip condition at the bottom surface and zero shear stress condition at the free channel surface with 5 % turbulent intensity.

4. CFD ANALYSIS AND FLUENT SETTINGS

Computational Fluid Analysis (CFD) is a branch of fluid mechanics that deals with numerical simulation methods and makes use of different algorithms to solve and analyze the fluid flow problems.

Fluent requires various settings like pressure velocity coupling, discretization schemes and relaxation factors. These important parameters are mentioned in Table 2.

Table 2: Solver settings and model relaxation factors in fluent

Pressure-Velocity Coupling	SIMPLE
Discretization of Gradient	Green Gauss Node Based
Discretization of Pressure	Second Order
Discretization of Momentum	Second order Upwind
Discretization of Turbulent Kinetic Energy	Second order Upwind
Discretization of Specific Dissipation Rate	Second order Upwind
Pressure Under-relaxation Factor	0.2
Momentum Under-relaxation Factor	0.6
Modified Turbulent Viscosity Under-relaxation Factor	

5. RESULTS AND DISCUSSION

The Residuals curves, Pressure profiles, drag coefficient and lift coefficient curves for turbine are evaluated using Fluent 14.5 solver. It is clearly evident from fig.7 and fig. 11 that coefficient of lift (Cl) converges towards zero with increasing number of iterations for both the turbulent models the reason being that the hydrofoil NACA0018 has zero camber i.e. it is symmetrical thus it generates zero lift at zero angle of attack.

5.1. Results for One-Equation Spalart-Allmaras Turbulent model.

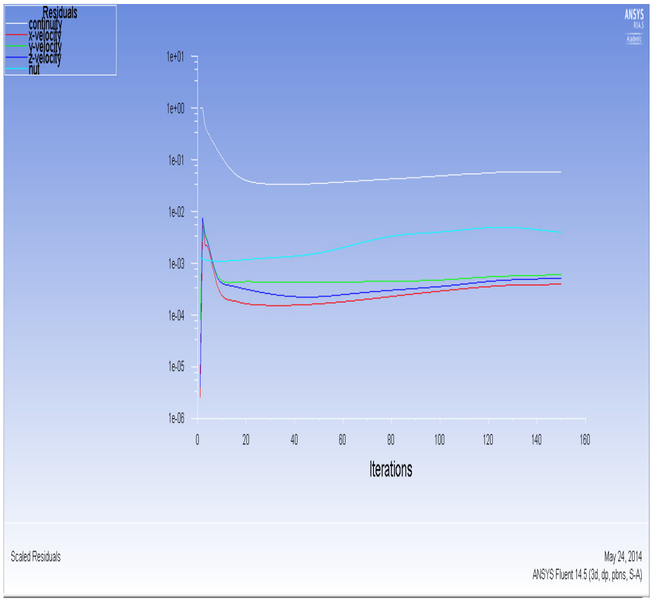

Figure. 5 Scaled residuals

The scaled residuals curve shows that the solution obtained is converging to exact solution and after nearly 20 iterations the curve is constant showing very small variation in the obtained solution.

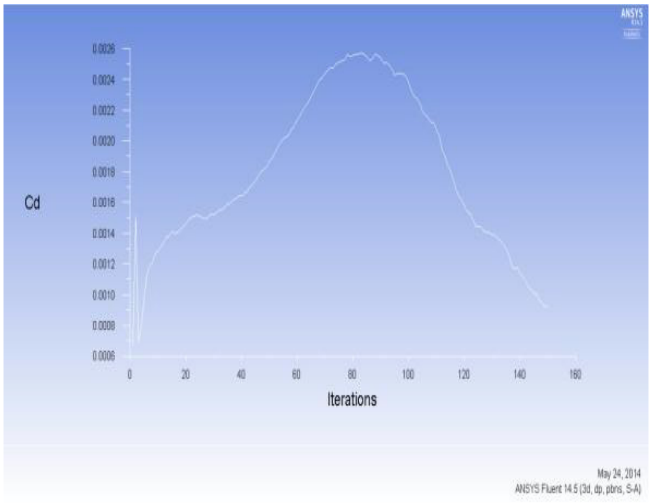

Figure. 6 Coefficient of Drag (Cd) convergence

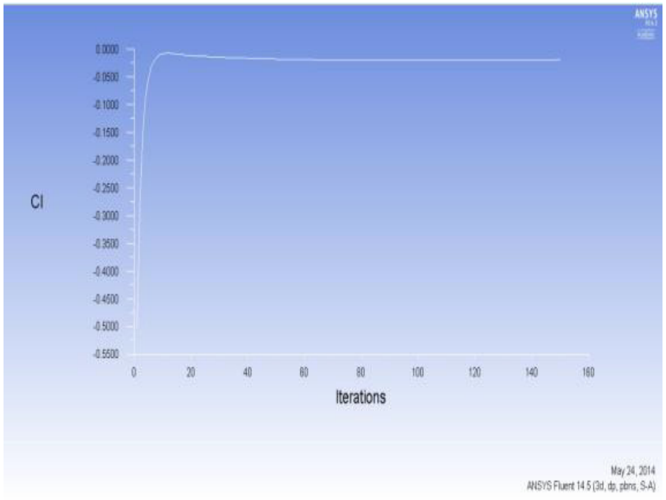

Figure. 7 Coefficient of lift (Cl) convergence

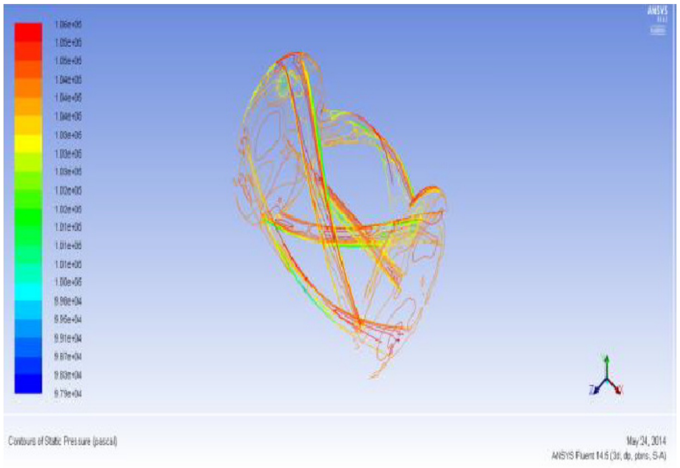

Figure. 8 Static pressure profile

Red region in figure.8 and figure.12 shows areas of maximum pressure region and similarly blue region shows profile of minimum pressure region. The results agree with the theory which states that the maximum pressure region in a turbine is along the thicker side and minimum pressure region is along the leading edge of the blade.

5.2. SST-kω model plots

The following results are for SST-Kw model which uses 2 equations to find the approximate results. The results are similar to that were obtained by Spalart-Allamras turbulence model.

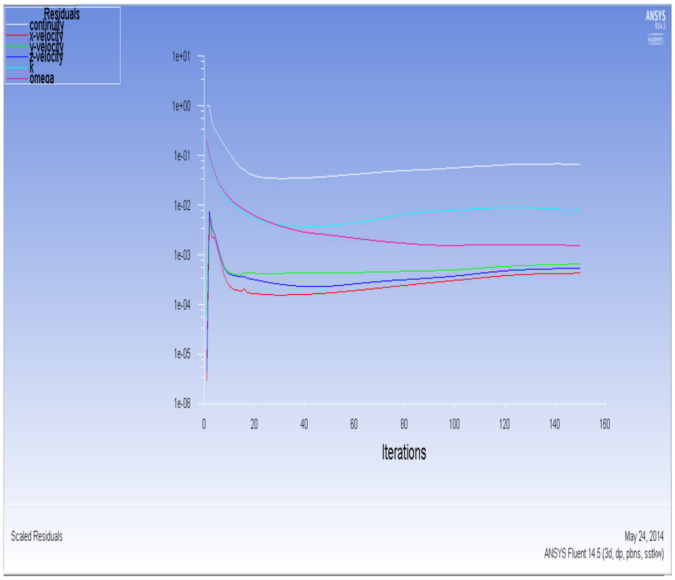

Figure. 9 Scaled residuals

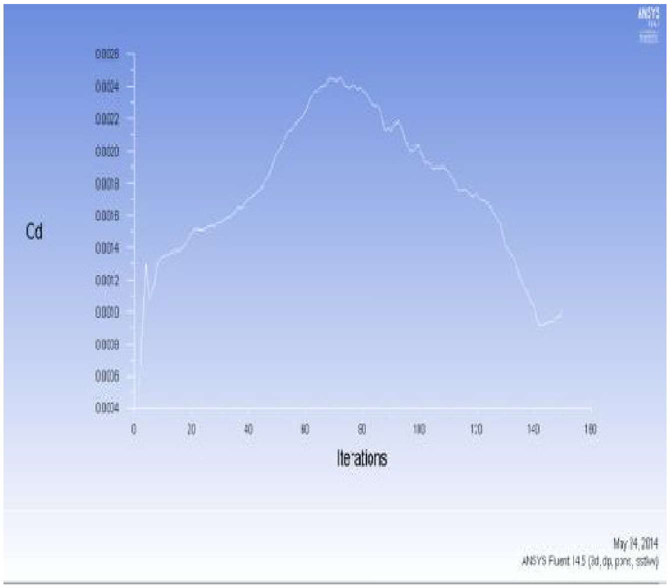

Figure. 10 Coefficient of Drag (Cd) convergence

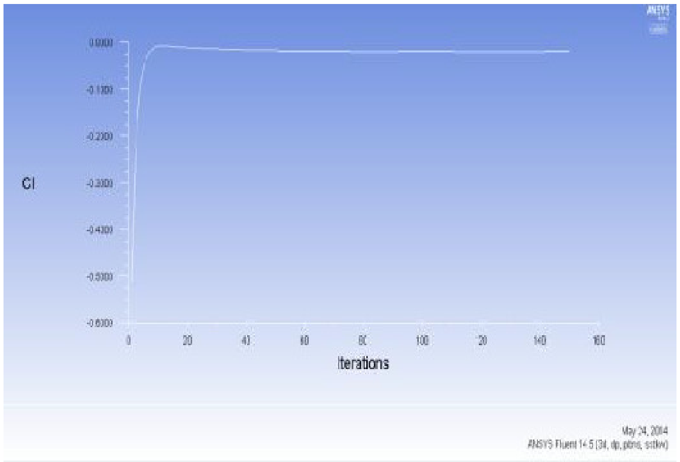

Figure.11 Coefficient of lift (Cl) convergence

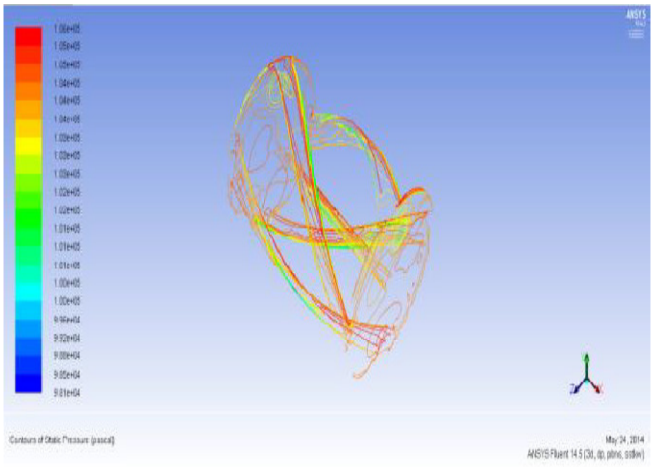

Figure.12 Static Pressure Profile

6. CONCLUSION

The present work demonstrates the simulation of pressure distribution around CFHT using CFD tool fluent v14.5, including underlying turbulence of fluid flow and also the viscous effects, without using tabulated drag and lift data. The primary objective of this research work was to develop an understanding of the pressure variation and profiles for the CFHT, as because of the limited literature available, this makes it very useful for novice in this field. The results obtained from both the models shows similar pressure variation and drag and lift convergence curves and are in accordance with the literature surveyed. However, there are a number of methods of analyzing the solution and obtaining the results, which can be implemented in future.

Also the paper draws attention to a new method of hydropower generation which doesn't require construction of large dams and tunnels to store energy. India has many perennial and seasonal rivers with huge hydropower potential; installing such turbines in place of conventional turbines will not only reduce project cost but will help prevent destructions of forests and villages.

7. ACKNOWLEDGEMENTS

The authors are thankful to Graphic Era University for providing modeling and computational software which were basic need of this project.

REFERENCES

[1] Oliver Paish, "Small hydro power: technology and current status" Renewable and Sustainable Energy Reviews 6 (2002) 537–556

[2] Energy Alternatives India (EAI), http://www.eai.in/ref/ae/hyd/hyd.html

[3] http://www.gcktechnology.com/GCK/pg2.html

[4] M.R. Castelli and E. Benini. Effect of Blade Inclination Angle on a Darrieus Wind Turbine. Journal of Turbomachinery-Transactions of the ASME, 134(3), (2012).

[5] Taylor Jessica Hall: Numerical simulation of a marine cross flow hydrokinetic turbine, University of Washington, (2012)

[6] Adam L. Niblick: Experimental and Analytical Study of Helical Cross-Flow Turbines for a Tidal Micropower Generation System, University of Washington, (2012)

[7] J. Zanette, D. Imbault, A design methodology for cross flow water turbines Laboratoire SolsSolides, Structures – Risques (3S-R) Domaine Universitaire, B.P. 53, 38041 Grenoble Cedex 9, France (2009)

[8] Claudio A Consul: Hydrodynamic Analysis of a Tidal Cross-Flow Turbine, Worcestor College, D.Phil, Trinity (2011)

[9] Samantha Jane Adamski: Numerical Modelling of the Effects of a Free Surface on the Operating Characteristics of Marine Hydrokinetic Turbines. University of Washington (2013)

[10] Menter: Zonal Two Equation kappa-omega Turbulence Models for Aerodynamic Flows (1993)

[11] http://airfoiltools.com/airfoil/details?airfoil=naca0018-il

[12] M. Shiono, K. Suzuki, and S. Kiho, Output characteristics of Darrieus water Turbine with helical blades for tidal current generations. Proceedings of the International Offshore and Polar Engineering Conference, 12:859 {864, 2002.}