Numerical Analysis of Crosss Flow Hydokinetic Turbine by Using Computational Fluid Dynamics

Prashant Gunai, Ajinkya Bhonge, Kaushal Joshi

LTCOE, University Of Mumbai, India

***Abstract*—*The invention of cross flow turbine industry from straight blades of the Darrieus turbine was modified by Alexander Gorlov into helical shape. There have been several research projects dealing with the design and analysis for tidal applications. This paper deals with the Numerical 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 15. Solidworks 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.*

Keyword*—*Naca0018, hydrofoil, Spalart-Allmaras, sst-k, CFHT.

I. INTRODUCTION

Cheap and efficient manner is the main motive of today's generation of electricity in every country. Each country develops the best possible method of electricity and renewables have become the primary source of power generation with the rapid consumption of conventional fuel. The flow potential of water currents in rivers, oceans, estuaries is thus studied as an immaculate and environmentally amiable source. Hydroelectricity is electricity produced from hydropower. In 2015 hydropower generated 16.6% of the world's total electricity and 70% of all renewable electricity, and was expected to increase about 3.1% each year for the next 25 years. There is a huge demand renewable form of energy. Gorlov turbine provides that alternative. Turbine rotates at twice speed as of water flow and in the same direction independent of water flow direction. The turbine is can be operated horizontally and vertically that reduces its construction and can be used widely in any open water flow along with shaft and generator arrangement. Hydrokinetic turbines are in early stage of developed on the reference of wind turbine theory.

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 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 model of height 1500 mm, 4 bladed turbine with 58.49º inclination angle. Pressure contours, drag and lift coefficient are calculated under a steady flow of 1.5 m/s using CFD solver fluent 15. 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 200 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 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 partsof 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 orientation of the hydrofoil geometry at different positions and angles.

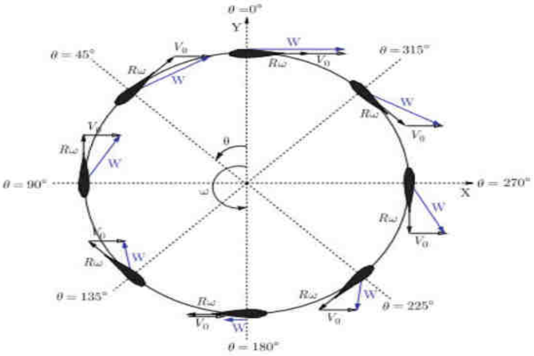

Fig.1: Turbine Positioning

II. 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. A reference model, a spherical 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

Fig.2: 3D CAD model

In figure '3' turbine is placed inside the channel which was developed in ANSYS 15 workspace .It is used as a duct through which water at normal atmospheric conditions will flow, thus water act as a continuum

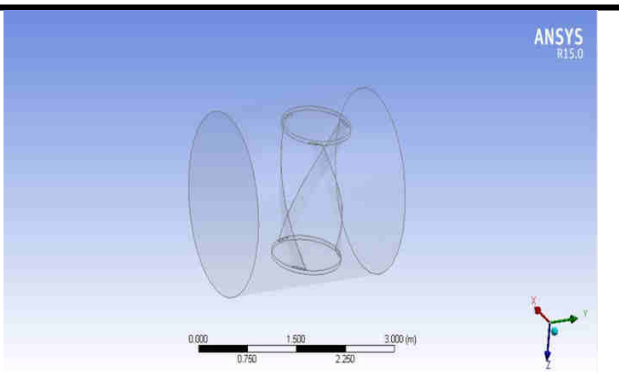

Fig.3: Turbine inside the channel

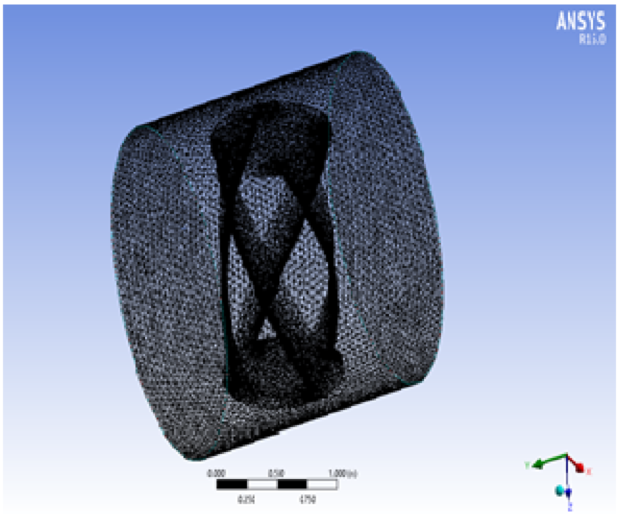

Fig.4: Meshing of the turbine

The CAD model of the turbine consists of four helical blades, two circular plates each 50mm thick and a central shaft that would be coupled with generator. The meshing was done with fine relevance center as shown in fig. 4. The meshed model consists of 3, 27, 697 nodes and 2, 48, 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. In general, the close the inclination angle is to 90º the better is the efficiency of the turbine**.**

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	274.889mm

4	Diameter	1000mm
5	Height of turbine	1500mm
6	Inclination angle	58.49º
7	Aspect ratio (A.R)	1.5
8	Solidity Ratio (σ) for 4 blade	0.35
9	Plate thickness	50mm

III. 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.

IV. CFD ANALYSIS AND FLUENT SETT INGS

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: Simulation parameters

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

V. RESULTS AND DISCUSSION

The Residuals curves, Pressure profiles, drag coefficient and lift coefficient curves for turbine are evaluated using Fluent 15 solver. It is clearly evident from fig.6 and fig.7 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.

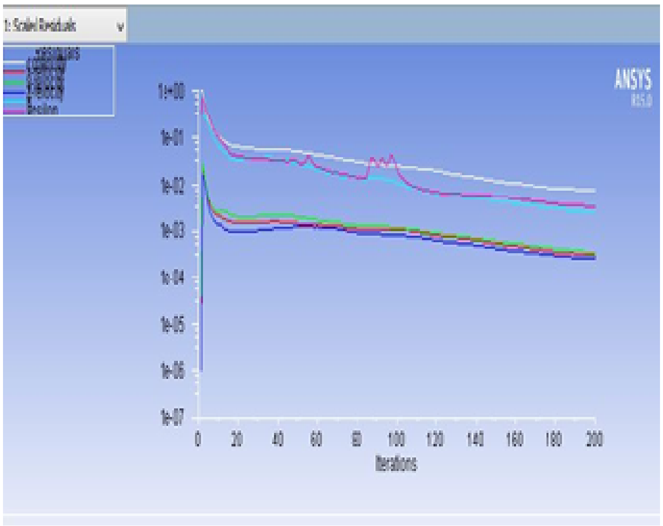

Fig.5: Scaled residuals

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

Fig.6: Coefficient of Drag (Cd) convergence

Fig.7: Coefficient of Lift (Cl) convergence

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 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-k_ model plots

The following results are for SSTequations to find the approximate results. The results are similar to that were obtained by Spalart model.

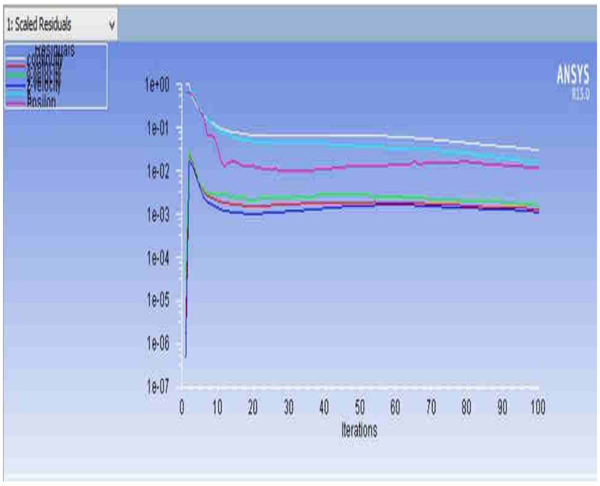

Fig.8: scale residual

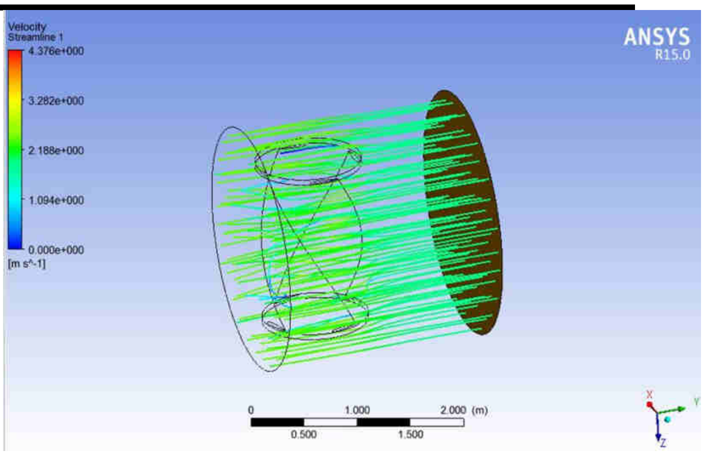

Fig.9: Velocity distribution

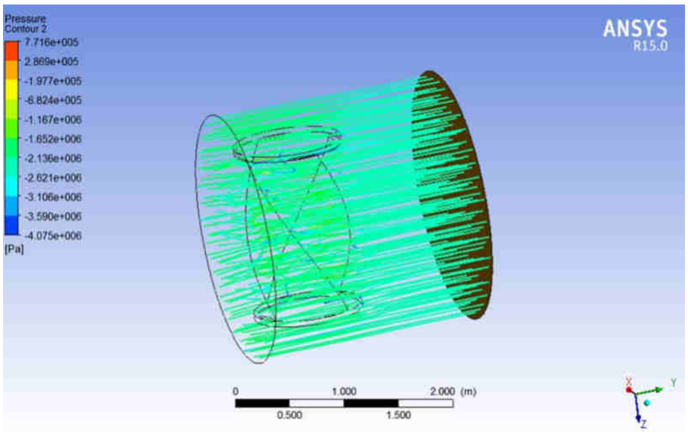

Fig.10: Pressure Distribution

Figure 9 & 10 represents velocity and pressure distribution of the water particle along the path. It also gives range of maximum velocity and pressure to minimum.

VI. CONCLUSION

The current work illustrate the simulation of pressure distribution around cross flow hydrokinetic turbine using CFD tool fluent 15, 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. 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. 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.

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] Taylor Jessica Hall: Numerical simulation of a marine crossflow hydrokinetic turbine, University of Washington, (2012)
[5] Adam L. Niblick: Experimental and Analytical Study of Helical Cross-Flow Turbines for a Tidal Micropower Generation System, University of Washington, (2012)
[6] 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).
[7] http://airfoiltools.com/airfoil/details?airfoil=naca0018-il
[8] Menter: Zonal Two Equation kappa-omega Turbulence Models for Aerodynamic Flows (1993)