AN INVESTIGATION OF AXIAL HYDROFOIL TIDAL TURBINE

Sharanya Balki[1]

[1]*Assistant Professor Department of Civil Engineering, R.M.K. Engineering College, Chennai, Tamilnadu, India*

Abstract -*Ocean energy uses the ocean's powerful currents, tides and waves to generate the total global electric power. The earth's tides caused by the gravitational force and the currents created by the winds are sources of renewable power that is free, reliable and predictable years in advance. By virtue of the basic physical characteristics that accrue to sea water, namely, its density (832 times that of air) and its non – compressibility, makes it unique when compared to the other renewable like wind. Early civilization though developed devices to convert tides into mechanical energy, the technology to cost effectively convert ocean waves, sub-surface currents and tidal flow into electrical energy is still in its early stages. This study was intended to have a better understanding and knowledge on the working of an axial tidal turbine. The work mainly focused on initial experimental tests on an axial hydrofoil turbine model. Preliminary results demonstrated that the turbine model could not take up any rotation because of improper orientation of blades, lack of streamlined shape of the blade profile. As a result, a parametric study was taken up to analyze the various aspects of the turbine using a commercial computational fluid dynamics solver, FLUENT. Two dimensional model of airfoil NACA 0018 was created, drawn and meshed in Gambit using the geometrical data gathered from National Advisory Committee for Aeronautics for the cross-section. The airfoil section's lift and drag co efficient from numerical study was compared with experimental data from literature in order to validate the CFD study. Comparative torque analysis was taken between the NACA profiles. Next simulations were carried out for two dimensional model of NACA 0024 profile. The performance was predicted by careful selection of the solidity and tip speed ratio*

Key Words: Tidal Current, NACA Profiles, Torque, Tip Speed Ratio, Solidity

1. INTRODUCTION

The oldest technology to harness tidal power for energy extraction involves building a dam or barrage, across a bay or estuary that has large difference in elevation between high and low tides. Though they have proven very durable, the barrage type of power plants is very expensive to build and is fraught with environmental problems from the accumulation of silt within the dam catchment area. Accordingly, engineers no longer consider barrage style tidal power feasible for energy generation. Over recent years, since major sites are already exploited or unavailable due to other considerations, use of small hydropower schemes like that of turbines is of increasing interest. The turbine technology is still in its early days and no full-scale commercial design and conditions has yet been tested and proven.

1.1 Need for Study

Owing to acute energy crisis that most developing countries including India are facing today, the interest in alternative energy sources has increased manifolds in the recent past and the potential of tidal energy, as a source of alternative energy perhaps cannot be underestimated. Ocean energy, including wave and tidal current energy, have the potential of playing a major role in the electricity market, providing reliable and sustainable energy. This non – polluting and non – toxic energy source will go a long way in solving our energy requirements. In order to make full use of the resources of tidal energy, the hydrokinetic conversion schemes are made use of. This mode of extraction of energy is in its early stages of development. This work aims at testing a physical model of a turbine with a particular blade profile scheme, and analyzes the characteristics features that mainly govern the efficiency of it. Knowledge of turbine efficiency limits helps to optimize design of hydro power farms.

1.2 Objective of Study

The main aim of the study is to explore the efficiency of the axial turbine device in a range of flow condition. To vary the aspects of the device configuration in order to improvise and examine the performance.

2. METHODOLOGY

Initial experimental analysis was carried out for the helical turbine, from the observations it was inferred that the turbine did not take up any rotation when it was completely immersed in the water. In order to improvise the model, a parametric study was carried out using solver FLUENT on various types of NACA profiles. A torque comparison was also taken up among the NACA profiles and the wedge profile of helical blade. Unsteady state analysis was also carried out on the chosen NACA profile

3. RESULTS AND DISCUSSION

The experimental analysis was conducted on 105 mm diameter with three bladed helical model at Centre for Water Resources Hydraulics Laboratory. The dimension of the flume is given by length 29m, width 1.5m, depth of water 0.7mand discharge 0.2 m^3 /s.Trial flume tests were taken up to investigate the maximum velocity that can be reached by contracting the cross section. Using the specific energy relationship the corresponding values were obtained. The values of Y_1 = 0.6 m and the corresponding values of Y_2 and V_2 were obtained. The discharge is taken about 0.2 m^3/sec. Minimum width = 0.25 m. B_1 = 1.5 m and B_2 = 0.25 m,B_1 = 1.5 m and B_2 = 0.3 m,B_1 = 1.5 m and B_2 = 0.35 m,B_1 = 1.5 m and B_2 = 0.4 m were obtained

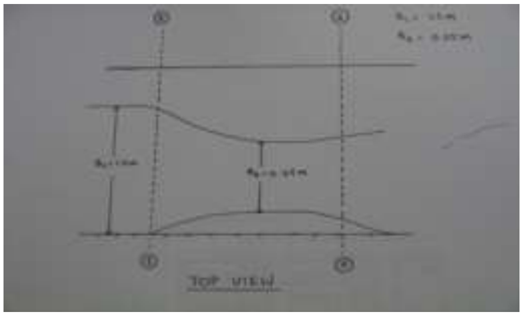
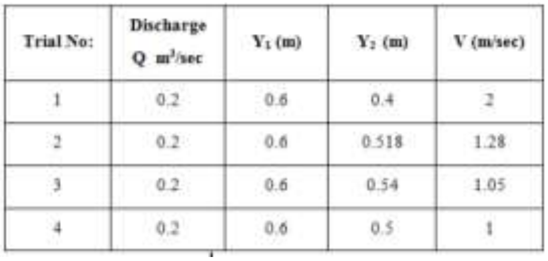


Fig -1: **Top View of Channel**

Table-1 Flow Conditions



Trial No:	Discharge Q m^3/sec	Y_1 (m)	Y_2 (m)	V (m/sec)
1	0.2	0.6	0.4	2
2	0.2	0.6	0.518	1.28
3	0.2	0.6	0.54	1.05
4	0.2	0.6	0.5	1

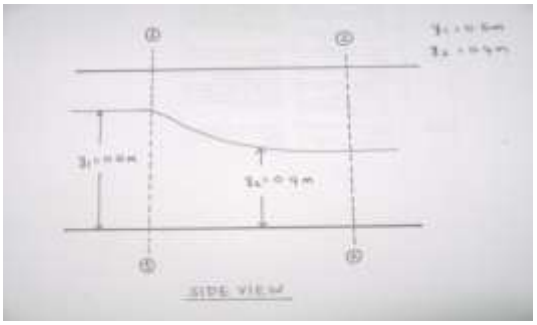


Fig -2: **Side View**

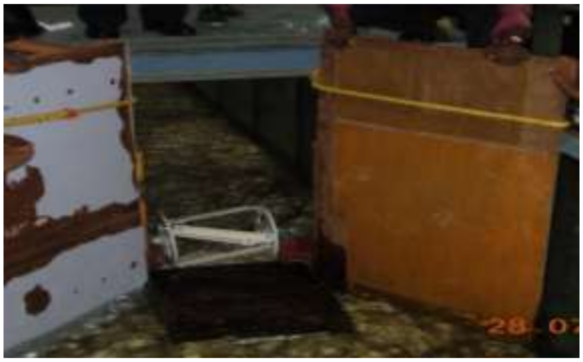


Fig -3: **Scheme of the Model**

The turbine failed to rotate when completely immersed in water. When the turbine was turned in the clockwise direction by an external push, it took up one half rotations and came to a halt .In anticlockwise direction, it sprang back to its original position. The possible reasons would be that geometry of the blade was hand twisted and the orientation was not proper and also the wobbling of the shaft affected the flow. This would have generated an impulse force on blade at downstream. Hence this led to the parametric study using FLUENT.

3.1 Preliminary CFD Studies On Wedge Shape Profile In Static Condition

The most fundamental consideration in CFD is treating a continuous fluid in a discretized fashion. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion. CFD modeling of turbine was performed using FLUENT. The computational flow analysis was performed on wedge shaped profile .The geometry of the turbine was constructed and the detailed specifications of the profile, length and number of blades are taken as, diameter- 0.300m, number of blades – 3, length - 0.275 m. A full model was constructed, containing the turbine blades and the shaft, and a background structured mesh. The total number of mesh faces that was generated was about 118735. The coarse mesh around the blade profile and shaft were created. A k – omega turbulence model was taken up. Boundary conditions adopted included a velocity magnitude of 1 m/s, X Component of flow direction =1, Y Component of flow direction = 0. The plot view shows the cross sectional position of the blades at the 0^o , 45^o 90^o position. Plot indicates the development of wakes around the blades A,B and C.

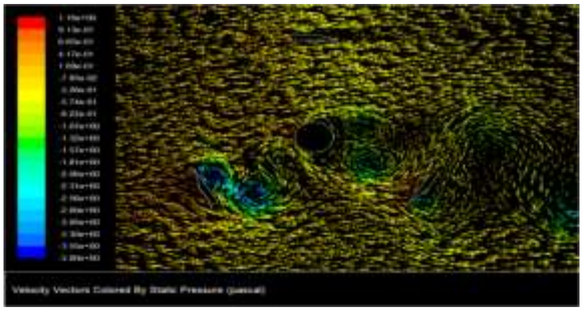


Fig -4: **Plot at Angle 0°**

The behaviour of the blades at 45° angle clearly indicates that the rotation is in the anticlockwise direction. Blade C is subjected to large amount of wakes, in turn reducing the amount of total torque when compared to the previous position

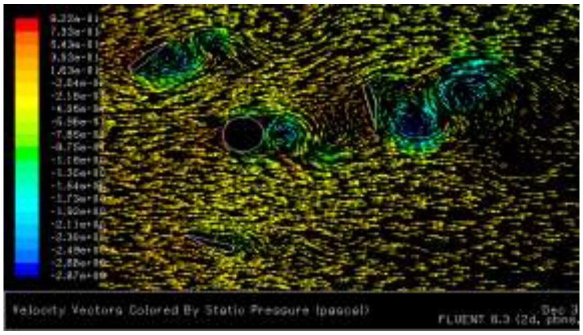


Fig -5: **Plot at Angle 45°**

The estimated static torque obtained for various angles of attack for a wedge shape .The torque takes up a cyclic pattern for every 1200 as blade 1 is replaced by blade 2 and blade 2 by blade3.

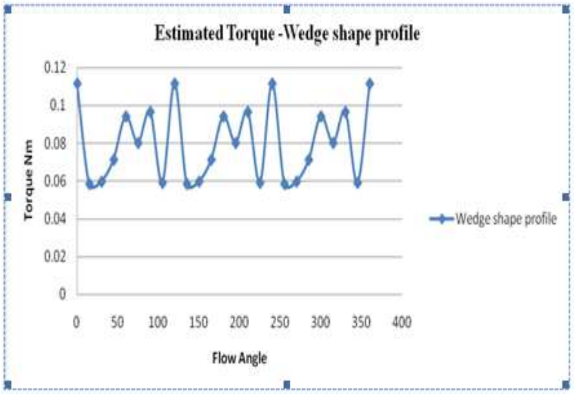


Chart-1 Estimated Static Torque- Wedge Shape Profile

3.2 Comparision Of Lift And Drag Characteristics between wedge and NACA 0018

Preliminary study on wedge shaped profile was followed by CFD analysis on NACA profile. Next part of the work included the comparision of the lift and drag coefficients for a particular profile, i.e. NACA 0018 in order to validate the CFD procedure. These symmetrical airfoils are comparatively easy to design and mainly used in vertical axis turbine. The model and mesh generated in gambit modeling software were read into the commercial CFD code, FLUENT 6.2.30 for numerical iteration. Boundary conditions adopted included velocity magnitude -1m/s, X – Component of flow direction =1, Y – Component of flow direction = 0. K-omega model specifies the turbulent flow model. The main feature of this model is that it is more accurate and reliable for a wider class of flows (e.g., airfoils, transonic shock waves). Fluid medium with density - 998.2 kg/m^3 and viscosity - 0.001003 kg/m-s were used. Simulation was carried on for a steady state condition i.e. analyzing the forces on the blades at fixed positions. Lift and drag co efficient for various angles of attack are monitored

Chart-2 Comparison of Lift Coefficients

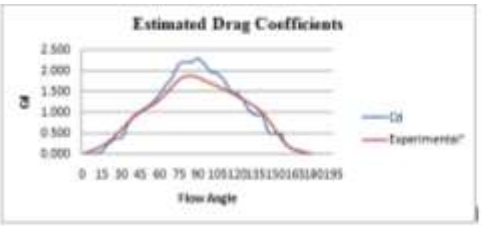


Chart-3 Comparison of Lift Coefficients

3.3 CFD Analysis For NACA 0018 and NACA 0024

Preliminary study on wedge shape profile was followed by CFD analysis on NACA profiles namely, NACA 0018, and NACA 0024. The primary difference between the three shapes is the cord to thickness ratio. Increasing in thickness assist airfoils to increase their lift force but at the same time drag also increases. The corresponding NACA profile torque values were estimated and compared with the values of wedge shape profile. The geometry of the turbine was constructed and the detailed specifications of the profile, length and number of blades are taken as number of blades – 3, diameter – 0.300 m, length - 0.275 m. A full model was constructed, containing the turbine blades and the shaft, and a background structured mesh. A k – omega turbulence model was taken up. Boundary conditions adopted included velocity magnitude 1 m/s, X Component of flow direction =1, Y – Component of flow direction = 0. The corresponding torque was estimated for various angles of attack .Since all the three blades were replaced at every 120^0 position, the resulting torque had taken up a cylic pattern at every 120^0 position. From the comparision between the wedge, NACA 0018, NACA 0024 profiles it clearly indicated that the NACA 0024 produced higher torque when compared with the wedge and NACA 0018 profie.

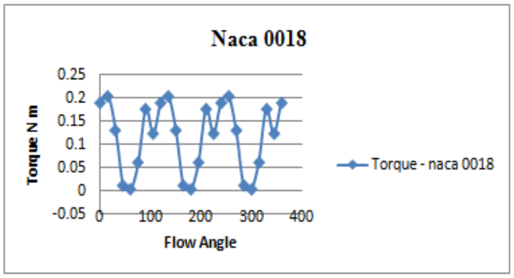


Chart-4 Static Torque (NACA 0018)

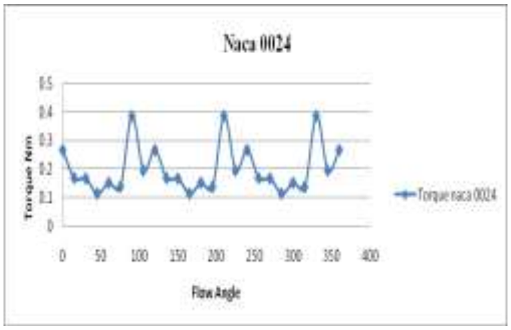


Chart-5 Static Torque (NACA 0024)

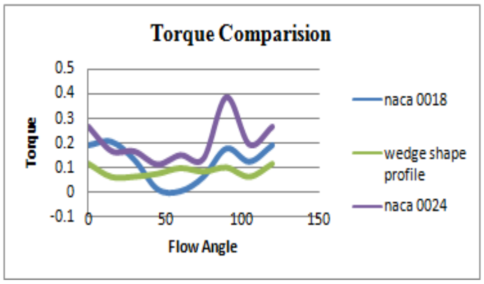


Chart-6 Torque Comparison

3.4 Transient Simulation with Moving Mesh Method For Naca 0024

NACA 0024 has been chosen for the unsteady state analysis with reference to results of high torque in steady state analysis. Considered case had a chord length of 18.5 mm and a three bladed turbine. Boundary conditions employed in 2 – dimensional computations consist of velocity inlet on left, outflow on right and two stationary walls at the top and bottom. Profiles representing blades are in inner part of a rotating domain which is separated from a steady domain (outer zone) by two sliding interfaces (rotor interface and stator interface) specified as a boundary condition of type sliding mesh. In this case, computational domain consists of a rotating zone (zone with 3 blades) and a steady zone, which includes water environment outside the rotating zone. Turbine was set up to a diameter of 0.3m. The inlet width was about 3 times the diameter of turbine and the outlet about 4 times the diameter. The model and the mesh generated in gambit modeling software were read into commercial CFD code, FLUENT for numerical iterative solution. The RANS equations were solved using the green gauss cell based gradient option and sliding mesh method was used to rotate the sub – domain of turbine blades. Initial steady state analysis was stabilized. After stabilizing the flow, dynamic analysis was done using the sliding mesh technique.

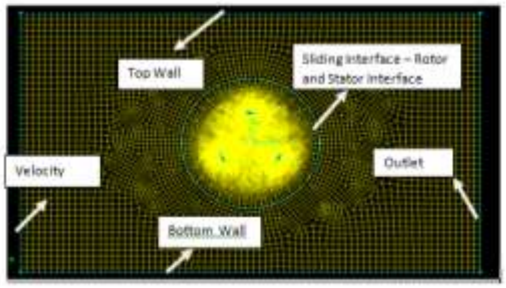


Fig -6: **Overall View of the Mesh**

After about 100 iterations without the rotation of the turbine blades, the steady flow field was developed. Then, the sub-domain was let spin and the unsteady flow computation was performed. The time step size of 0.0069444 second was used with 100 iterations per time step. The computation was continued until the stable periodic solutions were obtained. Time step size was set corresponding to 2.5 degree for each rotational speed of the rotor (ω) corresponding to each TSR. Flow condition for various tip speed ratios and the corresponding speed of the turbine are tabulated

Table-2 Flow Condition

TSR	Free Stream Velocity (m/s)	Angular Speed of Turbine (m/s)	N(rpm)
0.2	1.2	0.24	15
0.78	1.2	0.936	60
0.9	1.2	1.08	69

Whenever the turbine angular motion goes to a higher value, it produces a lower torque and ultimately ends up with low power generation. At TSR -0.78 the turbine had generated power about 0.77watts which is extremely low. If the TSR is low then it indicates that the turbine becomes inefficient, because a lot of flow particles pass through turbine without losing their kinetic energy. If the TSR is high then it indicates the better efficiency of the turbine because the flow particles that do not touch any blades and will sweep away the particles that have lost most of their kinetic energy to the turbine and this will increase the overall efficiency of the turbine. In order to improvise the present condition, the existing model was scaled up to a dimension of 0.75 m diameter of turbine, chord length – 0.046 m, number of blade -3, blade profile –NACA 0024. The same analysis was carried out for the scaled up turbine model indicated the results of the simulation taken for various TSR – 0.2, 0.78 and 0.9 and the corresponding torque and the power values.

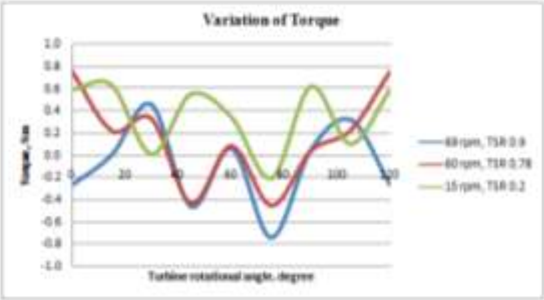


Chart-7Torque Comparison with TSR (NACA0024)

The graphs clearly indicated that the maximum efficiency appeared to be obtained around TSR of 0.78 and the corresponding power about 1.68watts. It is confirmed that a proper combination of the solidity and TSR is essential for the optimum efficiency of the turbine.

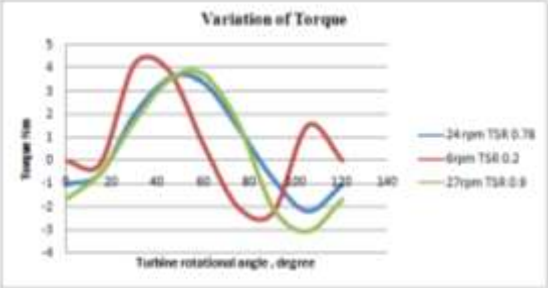


Chart-8Torque Comparison with TSR(Scaled up model NACA 0024)

higher the solidity the efficiency drops. It is should be limited by moment of inertia. As s result, the performance of the turbine models depends on various factors like free stream velocity, type of blade profile,,solidityaspect.

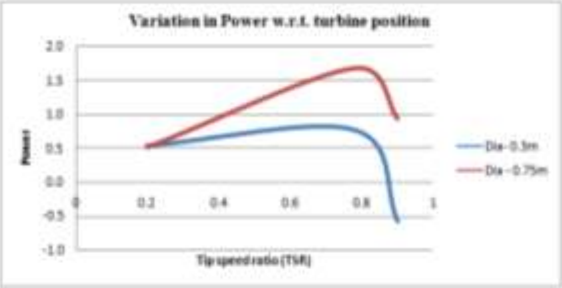


Chart-9 Power Comparision

4. CONCLUSION

Experimental investigation on the helical blade model clearly indicated that the turbine model did not take up any rotation, reason being the lack of streamlined profile, improper orientation of blades, and the free stream velocity of 1m/s. The CFD analysis clearly indicated that the performance of NACA 0024 profile was better when compared to the wedge shaped profile. The estimation of the hydrodynamic forces on NACA 0018 showed good agreement with that of the literature experimental results thus validating the CFD study. Comparing the graphs of the transient simulation of NACA 0024 of both the turbine models (chord length 0.0185 m and chord length 0.047 m), it clearly indicated that the scaled up model was able to generate better power. Performance of a turbine model greatly depends on parameters like the free stream velocity, type of blade profile and solidity aspect. Higher solidity - efficiency drops .The solidity ratio should be limited.

REFERENCES

- G. Alexandar (1998), 'Development of the Helical Reaction Hydraulic Turbine' – Final report submitted by The US Department of Energy.
- H Beri and Y.Yao. (2010), 'Numerical Simulation of Unsteady Flow to Show Self-starting of Vertical Axis Wind Turbine Using Fluent', accessed on Januray 01, 2001
- D.P Coiro., U. Maisto ., and F. Grasso . (2006), Report on 'Horizontal Axis Tidal Current Turbine: Numerical and Experimental Investigation', University of Naples.
- M.S.U.K Fernando. and Modi V.J. (1989), 'A Numerical Analysis of the Unsteady Flow Past Savonious Turbine', *Journal of Wind Energy,* Vol. 32, pp 303-327.
- M.J Khan M.J., Bhuyan G., and Iqbal M.T. (2009), 'Hydrokinetic Energy Conversion Systems and Assessment of Horizontal and Vertical Axis Turbine for Tidal Applications: A Technology Review', Journal of Applied Energy, Vol.5, pp 1823-1835.