NUMERICAL AND EXPERIMENTAL INVESTIGATION OF RIVER HYDROKINETIC TURBINE FOR WATER PUMPING APPLICATION

Mohammad Fozan ur Rab¹, Muzammil Ejaz³, Fahad Qureshi⁴, Muhammad Hassan⁵  
NED University of Engineering and Technology, Karachi, Pakistan

Wajiha Rehman²  
University of Engineering and Technology, Lahore, Pakistan

ABSTRACT

This study is carried out to design a low head turbine which will utilize the kinetic energy of flowing river stream. This turbine will be used to drive a hydraulic pump by coupling the turbine’s shaft with the pump’s shaft. In this way, this system requires no external power and heavy construction work. Axial flow Hydrokinetic turbine’s design is chosen because it has high efficiency and fabrication ease. A scale-down model of turbine is fabricated and tested in a Circulating Water Channel (CWC) at 0.7 m/sec flow velocity. Based on experimental work, a CFD model is designed. The turbine’s model is analyzed by using Computational Fluid Dynamics (CFD) at same inlet conditions and then numerical value of moment is compared with the experimental value at same rotational speed. Both values of moments are very close to each other which validates the CFD model. The k-ω SST turbulence model is utilized to close the system of equations. The computational grid contains approximately 5 million cells with average y+ value less than five. In future, this validated CFD model will be used to simulate the full-scale turbine prototype in actual river conditions and will be helpful in optimizing the final design.

INTRODUCTION

Renewable resources are capable of meeting energy demands while decreasing the effect of global warming¹. Water is a dominant renewable energy resource that is supplying 18% of global electricity. The approximated global gross, economic, technical and exploitable hydropower potential is estimated as 128, 21, 26 and 16 Penta watt hours per year, respectively². According to the National Electric Power Regulatory Authority (NEPRA) report³, Pakistan’s hydel potential is 40,000 MW but only 15% of this potential is exploited. A huge potential is available in the form of low head e.g. canals, run of rivers and farm channels where conventional turbine systems cannot be used. Hence, a novel technology must be used to exploit this potential.

Currently, the two major methods to produce electricity from water are hydrostatic method and hydrokinetic method. In hydrostatic method, dams and other civil structures are used to store water and then its energy is extracted by turbines⁴. In hydrokinetic system, the kinetic energy of the moving fluid is converted into electricity⁵. Globally, various studies have done to examine the working and designing parameters of hydrokinetic turbine by using various approaches. Hydrokinetic turbine system is a popular research topic, which intrigued engineers from all over the world. Numerical tools like Computational Fluid Dynamics (CFD) is one of the most powerful tools that has revolutionized the designing procedure. Many researchers, from all over the world,
are using CFD to design hydrokinetic turbines. Kolekar et al.\textsuperscript{6-8} and Mukherji et al.\textsuperscript{9} optimized the design of a 12 kW-horizontal axis turbine by employing CFD and blade element theory. Batten et al.\textsuperscript{10-12} used Blade Element Momentum model (BEM) to design a horizontal axis turbine for the exploitation of tidal energy. They also performed experiments in a cavitation tunnel by using a scaled model and found that BEM model satisfied experimental results. Researchers\textsuperscript{13-15} improved BEM model to consider the effects of dynamic stall, rotational flow and losses at the tip of blade. Later, three-dimensional inviscid models were introduced to explain the hydrodynamics of turbine in more detail than the BEM model. Researchers\textsuperscript{16-18} improved 3D inviscid model by introducing lifting line, panel and vortex lattice method but it was incapable of taking the effect of viscous forces that are required for accurate performance prediction. Tian et al.\textsuperscript{19} designed and performed CDF analysis of a Horizontal Axis Water Turbine (HAWT) to power the Underwater Moored Platforms (UMP). They used ANSYS FLUENT v13.0 as CFD tool and selected k-\omega Shear Stress transport (SST) model to solve the Reynold’s Averaged Navier Stokes (RANS) equations. The results of the simulation were validated by experimental data and which supported their numerical model. Lawson et al.\textsuperscript{20} optimized the design of a Horizontal Axis Tidal Turbine (HATT) and employed k-\omega SST turbulence model to get results for the turbine’s performance. The results obtained were compared with BEM and there was a reasonable agreement. Nak et al.\textsuperscript{21} also designed a HATT and performed experimental and CFD analysis by using ANSYS CFX v13.0. Authors used k-\omega SST turbulence model and plot the performance characteristics curves of the rotor and found the CFD results quite accurate. Various other researchers\textsuperscript{22-24} used k-\omega SST turbulence model for the prediction of turbine’s performance and found good agreement with the experimental data.

Myers and Bahaj\textsuperscript{25} performed experiments on horizontal axis water turbine to determine the designing parameters of rotor. They found that the stall delay of the foil greatly depends on the twist angle of blade, lift and drag performance, centrifugal force and rotor’s yaw angle. Hayati et al.\textsuperscript{26} found that the thrust performance of turbine increased by increasing the rake angle of the propeller. Singh and Nestmann\textsuperscript{27,28} did parametric study to optimize the geometry of rotor. By modifying the inlet tip of the runner to reduce discharge consumption, they improved efficiency i.e. from 55% to 74%. They modified the inlet and exit angles of the blades, for three different stages of runner, and found that was increased by decreasing the exit blade angle. Singh and Nestmann\textsuperscript{29} found designing parameters of low head horizontal axis water turbine. They found the relation of blade number with turbine’s performance and concluded that blade number is more significant than blade’s height to achieve high efficiency.

The work in this paper focuses on the experimental and numerical study of the scale-down model of a low head turbine system. The full-scale prototype will drive hydraulic pump to pump water from river to nearby town situated in Gilgit-Baltistan region of Pakistan. The residential area is at an elevation of 45.72 m from the river. The proposed site of reservoir tank is 518.7696 m away from river while the minimum depth of river is 1.524 m (variable in summer and winter). The river flow velocity reported in winter is slightly greater than 2 m/sec. According to initial study, two such type of turbines will be needed to drive two pumps of 2.2 kW each to fulfill the water requirement of households in that area. The detailed specifications of the proposed site are mentioned in Fig. 1.
After site survey and literature study, it has been deduced that hydrokinetic turbine is feasible for installation on site. The turbine would be installed on a floating platform with steel cable-ties support. The turbine will be fully submerged, and pump will be coupled with turbine’s shaft. The arrangement of turbine is shown in Figure 2.
HYDRAULIC DESIGN

In this paper, the design of runner is taken from Schleicher et al. \textsuperscript{30} and Riglin et al.\textsuperscript{31} works. The turbine’s rotor and diffuser geometry with solid cylindrical outer zone for Computational Fluid Dynamics (CFD) analysis was prepared by using Computer-Aided Design (CAD) software. The area Ratio (AR) of diffuser is about 1.09 and the meridional length ($\Delta m$) is 0.0190 m. The tip and hub diameters of turbine are $D_t=0.0844$ m and $D_h=0.0201$ m respectively while the diffuser inlet and outlet diameters are 0.1025 and 0.1118 m.

Figure 3. (a) Front view of rotor, (b) Side view of rotor (c) Side view of diffuser

Figure 4. CAD model of rotor with diffuser casing
EXPERIMENTAL ANALYSIS

Experimental setup

At first the experimental setup was designed to test the scaled model of turbine. So that, based on experimental results, a Computational Fluid Dynamics (CFD) model can be designed and its performance can be validated. The scaled model was 3D printed and then tested in water tunnel in Fluid Mechanics laboratory. The turbine and diffuser were printed by using Polylactic Acid (PLA) plastic material and then coated with synthetic paints to get smooth surface. PLA plastic is different from thermoplastic polymers because it is derived from renewable resources like cornstarch or sugar cane. The turbine was placed in a Circulating Water Channel (CWC). Width of the test channel was 0.3 m while water height was variable depending upon the flow rate of water. At a flow rate of 0.023 m$^3$/sec, the height of water in CWC was around 0.105 m and turbine was fully submerged in water. The velocity of water was around 0.73 m/s. Under these conditions, rotational speed of turbine was measured using a slow-motion video camera, turbine was rotating around 480 rpm without any external braking force (free run).

To calculate the power utilized by turbine at 480 rpm, an electrical Direct Current (DC) motor was used which was assumed to be 80% efficient. Turbine shaft was attached to the DC motor and DC was supplied. The rotational speed was measured by using a laser gun tachometer. At a voltage of 5.1 V and 0.1 Amp current, the turbine was running at 480 rpm. Using the relation in Eq. 7, 0.5 Watts of electrical power was obtained.

\[
P = I \times V \tag{7}
\]

\[
0.408 = (0.1 \times 5.1) \times 0.8
\]

As DC motor was 80% efficient, the mechanical power was calculated to be around 0.408 watts. The calculated electrical power was equated with mechanical power to calculate the torque. Using the relations given in Eq. 10, the torque generated by turbine at 480 rpm (50.26 rad/s) was calculated.

\[
P = T \times \omega \tag{8}
\]

\[
\omega = \frac{2\pi N}{60} \tag{9}
\]

\[
P = \frac{2\pi NT}{60} \tag{10}
\]

Here, \(P\) is power, \(I\) is current in amps, \(V\) is voltage, \(T\) is torque, \(\omega\) is angular velocity and \(N\) is revolutions per minute.
Table 2. Applied conditions on the test bench

<table>
<thead>
<tr>
<th>Input electrical power (P*)</th>
<th>Shaft power (P) (P* × 0.8)</th>
<th>Rotational Speed (N)</th>
<th>Torque (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Watts</td>
<td>Watts</td>
<td>rpm</td>
<td>N-m</td>
</tr>
<tr>
<td>0.51</td>
<td>0.408</td>
<td>480</td>
<td>0.00812</td>
</tr>
</tbody>
</table>

Figure 5. Experimental setup of turbine

Results of experimental analysis

Table 3. Results obtained during experiment in CWC

<table>
<thead>
<tr>
<th>Flow rate (Q)</th>
<th>Speed at free run (N)</th>
<th>Velocity (U)</th>
</tr>
</thead>
<tbody>
<tr>
<td>m³/sec</td>
<td>rpm</td>
<td>m/sec</td>
</tr>
<tr>
<td>0.023</td>
<td>480</td>
<td>0.7</td>
</tr>
</tbody>
</table>

NUMERICAL ANALYSIS

Computational Grid and Boundary Conditions

Rotating reference frame was used in the proximity of turbine to transform flow from inertial frame of reference to non-inertial frame of reference to capture rotational effects. The diameter of rotating
reference frame was kept slightly larger than $D_t$ and its length was also greater than the length of diffuser. Hexahedral dominant grid was utilized to discretize the flow domain using Finite Volume Method (FVM) into approximately 5 million cells. The grid that was generated comprised of varying degree of refinement levels in different regions of flow domain. The generated grid resulted in average $y+$ value of less than 5 along the walls of turbine and diffuser.

![Figure 6. Meshed model of turbine](image)

The length of the computational domain is 2.5 m while both height and width are 1 m each. The computations were performed in SimScale (Open FOAM based Cloud-simulation platform) using pressure-based segregated Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm. The uniform velocity inlet boundary condition of 0.7 m/sec was specified at the inlet of flow domain. The constant gauge pressure of 0 Pa was set at the domain outlet. The bottom face of domain was set to no-slip wall while the remaining faces were modeled as slip-wall with zero
velocity gradient. The moving reference frame angular velocity was set to 50 radian/sec corresponding to rotational speed of scaled model in experiment.

![Flow field with labelled boundary conditions](image)

**Figure 8. Flow field with labelled boundary conditions**

**Governing equations**

The flow is assumed incompressible, turbulent and steady. All thermo-physical properties of water are independent of temperature. Governing equations in cylindrical coordinates are written as,

**Mass conservation:**

\[ \frac{1}{r} \frac{\partial (u_r)}{\partial r} + \frac{1}{r} \frac{\partial (u_\theta)}{\partial \theta} + \frac{\partial (u_z)}{\partial z} = 0 \]  

(1)

**Momentum conservation:**

\[ \rho \left( \frac{\partial (u_r)}{\partial t} + u_r \frac{\partial (u_r)}{\partial r} + \frac{u_\theta}{r} \frac{\partial (u_r)}{\partial \theta} - \frac{u_\theta}{r} + u_z \frac{\partial (u_r)}{\partial z} \right) = -\frac{\partial P}{\partial r} + \rho g_r + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_r}{\partial r} \right) - \frac{u_r}{r^2} + \frac{1}{r^2} \frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2} \right] \]  

(2)
In equations, \( u_r, u_\theta, u_z \) are radial, tangential and axial components of velocity while \( g_r, g_\theta, g_z \) are radial, tangential and axial components of gravitational acceleration, respectively. The density of water is denoted by \( \rho \) while dynamic viscosity is denoted by \( \mu \).

**Turbulence Modelling**

Menter’s \(^{32}\) k-\( \omega \) Shear Stress Transport (SST) two-equation eddy-viscosity model was used to model turbulence. The k-\( \omega \) SST can model the transport of turbulent shear stress while accurately predicting the onset and amount of flow separation under adverse pressure gradient. It demonstrates k-epsilon type behavior in far field and is better at capturing free-shear effects while it switches to k-omega formulation in the near-wall region inside boundary layer. k-\( \epsilon \) model is usually not suitable for modelling large adverse pressure gradients while k-omega has its limitations in modelling free-shear flows (wake, recirculation). k-\( \omega \) SST model incorporates Bradshaw’s \(^{32}\) observation that the principal turbulent shear stress is proportional to the turbulence kinetic energy in the wake region of boundary layer by redefining the eddy viscosity formulation for adverse pressure gradient boundary layer. Hence, it is the most suitable model for the simulation of hydrokinetic turbine. The equations for turbulence kinetic energy and specific dissipation rate are written as

\[
\frac{\partial (pk)}{\partial t} + \frac{\partial}{\partial x_i} (pku_i) = \frac{\partial}{\partial x_j} (\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k
\]  

\[ \text{(5)} \]

\[
\frac{\partial (p\omega)}{\partial t} + \frac{\partial}{\partial x_i} (p\omega u_i) = \frac{\partial}{\partial x_j} (\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega
\]

\[ \text{(6)} \]
Here, \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity for \( k \) and \( \omega \). The turbulence kinetic energy generated by velocity gradient is represented by \( G_k \) while \( G_\omega \) represents the generation of \( \omega \). The dissipation in \( k \) and \( \omega \), due to turbulence, is represented by \( Y_k \) and \( Y_\omega \), respectively. \( D_\omega \) shows the cross-diffusion term.

**Results of numerical analysis**

The quantities of interests are Moment along rotational axis and axial thrust force experienced by the turbine due to water flow. The thrust force is approximately 1.2231 N while the Pressure moment in z-direction is found to be 0.00760336 N-m which is in good agreement with experimental data.

![Figure 9. Flow velocity streamlines and turbine surface pressure contours](image)

In figure 9, the streamlines of flow colored by velocity magnitude are plotted and it has been observed that the flow behind the turbine is highly rotational in nature and downstream wake is characterized by the region of high velocity deficit.

<table>
<thead>
<tr>
<th>Table 1. Results obtained from Numerical Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Upstream flow velocity (U)</strong></td>
</tr>
<tr>
<td>m/sec</td>
</tr>
<tr>
<td>0.7</td>
</tr>
</tbody>
</table>
Figure 10. Variation of pressure moment along rotational axis with simulation time

Figure 10 represent trend of pressure moment along rotational axis with the evolution of simulation time. As the simulation proceeds, the oscillations in moment value are diminished.

Figure 11. Contours of Static pressure

Figure 11 illustrates that the pressure is greatest near the shaft cone of the turbine. It is the major contributor to the thrust force experienced by shrouded turbine. The slight disadvantage of addition of diffuser is that it gives rise to additional axial thrust force.
The velocity contours plot in Figure 12 depicts that the region behind the turbine is dominated by strong wake and the diffuser section has resulted in increase in flow velocity in proximity of blades. The presence of diffuser results in improved efficiency and higher peak power coefficient achieved by the turbine.
From figure 13, it can be observed that the multiple vortices are shed from tip of the blades and the hub. The regions of high vorticity are slowly dissipated as they are convected downstream of the turbine. Smaller vortices are also shed from the diffuser and their strength is diminished much earlier than the strength of blades tip and hub vortices.

**Figure 14. Relation between Cp and TSR**

In Figure 14, the graph was plotted between the power coefficient and tip-speed ratio. The highest efficiency was achieved at TSR 2.4 which is 44%. By increasing the TSR beyond 2.4 and a decreased in efficiency was observed.

**Figure 15. Relation between \( C_T \) and TSR**

In Figure 15, a graph was plotted between thrust coefficient and tip-speed ratio to study the relationship of thrust coefficient with TSR. The highest \( C_T \) value was obtained at TSR of approximately 1 and decrease in \( C_T \) was observed with the increase in TSR.
CONCLUSIONS

The performance of Hydrokinetic turbine is quantified both experimentally and numerically during this study. The best efficiency point is obtained at TSR of about 2.4 corresponding to the design point of turbine. The results obtained from CFD are in good agreement with experimental values having deviation of 6.4%. Both numerical analysis and experimental investigation of the scale model of axial hydro-kinetic turbine conducted in laboratory have shown promising trends for development of the full-scale prototype. In future, this study of scale-down model will be used to optimize the design and test the performance of the full-scale turbine.

CONTACT

Mohammad Fozan ur Rab has completed four years bachelor’s degree in Mechanical Engineering from NED University of Engineering and Technology, Karachi, Pakistan. His research interests include applied Computational Fluid Dynamics (CFD) and low Reynolds number Aerodynamics.

Email: fozan.khan1994@hotmail.com

Wajiha Rehman has completed four years bachelor’s degree in Mechanical Engineering from University of Engineering and Technology, Lahore, Pakistan. Her research interests include water turbines and hydrodynamics.

Email: wajiharehman11@gmail.com

ACKNOWLEDGEMENTS

The authors are grateful to Mr. Ghulam Farooq (Senior Executive, Pak Suzuki Motor Company Limited), Mr. Syed Mehdi Ali Rizvi (Owner & Director of PCI Automotive) and Mr. Muhammad Affan (Undergraduate Student, Department of Electrical Engineering, NED University of Engineering and Technology) for their unconditional help and support.

REFERENCES


