CFD Analysis of Tidal Current Turbine Performance with Different Boundary Conditions

Saeed Badshah¹, Mujahid Badshah¹, Noman Hafeez¹, Sakhi Jan¹ and Zia Ur Rehman¹
1 Department of Mechanical Engineering, International Islamic University, Islamabad, 46000, Pakistan
E-mail: saeed.badshah@iui.edu.pk

Abstract. The outcome of Numerical methods such as Computational Fluid Dynamics (CFD) based on Reynolds Averaged Navier Stokes (RANS) is dependent upon selection of several critical parameters. The effect of boundary conditions, boundary flow modelling and turbulence numerics has been quantified in this study. Steady state RANS CFD simulation are performed by utilizing a Rotating Frame of Reference (RFR) model along with a Shear Stress Transport (SST) closure model. The simulated performance results were compared with experimental data over Tip Speed Ratios (TSR) from 2-9. Four different sets of boundary conditions that can possibly represent an experimental water channel were evaluated. The variation of boundary condition resulted in the variation of prediction error at different TSR’s. Critical variations were observed at the extreme operating conditions of minimum and maximum TSR’s. However, the $C_p$ prediction error at the optimum TSR was not much affected by the variation of boundary conditions.

1. Introduction
Fossil fuel resources are rapidly depleting and their use for energy generation is also threatening the global environment. Renewable energy sources are therefore widely researched through the world to replace the fossil fuel based energy generation. One of the most predictable renewable energy source with a significant global potential is to utilize the hydrokinetic energy of the tidal currents. These currents are mainly generated by the variation in lunar gravity and to some extents by the solar and earth gravity. The energy generated from the currents in tidal streams is commonly referred as tidal stream or tidal current energy. Two of the most widely used types of devices used for harnessing tidal current energy are the horizontal and vertical axis turbines. The blade profile for a Horizontal Axis Tidal current turbine (TCT) utilizes pressure difference across the two blade surfaces to rotate the turbine. There are commonly two types of numerical approaches used for the performance evaluation of TCT namely as blade element momentum theory (BEMT) and computational fluid dynamic (CFD). Blade element momentum theory uses two theories for the determination of turbine torque and thrust for analysis of the horizontal axis TCT [1-4]. Several studies have been reported in the recent literature to evaluate the performance of three dimensional (3D) TCT using CFD technique [5-
9]. There are several parameters that can affect the CFD solution like domain size, mesh, turbulence models, advection schemes, turbulence numeric and the choice of the boundary conditions. The effect of velocity shear on the performance and structural response of a turbine are investigated using a small-scale horizontal axis turbine with a rotor diameter of 0.5 m. A 3D Reynolds averaged Navier–Stokes (RANS) CFD procedure was developed and validated for performance prediction of tidal current turbine [10]. The effects of blockage ratio and boundary proximity on tidal turbine performance are quantified using computational fluid dynamics [11,12]. Velocity profiles in tidal channels cause cyclic oscillations in hydrodynamic loads due to the dependence of relative velocity on angular position, which can lead to fatigue damage. The effect of velocity profile on the load variation and fatigue life of large-scale tidal turbines is quantified. Due to the effect of velocity profile, the mean stress is decreased, whereas, the range and variation of stress are considerably increased [13]. Fluid Structure Interaction (FSI) models are used to model the hydroelastic behavior. The FSI approach was adopted to develop an FSI model for the performance evaluation and structural load characterization of a TCT under uniform and profiled flow [14].

In this research work the effect of those boundary conditions that could possibly represent a water channel on the performance prediction of TCT is analyzed. The objectives were therefore (1) to develop and validate a 3D RANS CFD procedure for the performance evaluation of a model scale TCT (2) to evaluate the impact of different sets of boundary conditions on the performance prediction accuracy of the CFD model.

2. Numerical modelling

The TCT rotor used for this study is a 1:40 scaled model of the RM1 TCT design developed by the U.S. Department of Energy (DOE). The geometric model consists of two bladed rotors with a rotor diameter of 0.5m. Detail of rotor design, experimental setup and experimental results can be found in [15]. A tetrahedral mesh scheme was adopted, with higher density around the blade surface and inner domain as compared to outer domain as shown in the Fig 1. The total number of elements were 3.9 million. To model boundary layer flow separation around the blade surface sixteen prism layers were added. The distance of the first layer from the blade surface was 0.067m with a growth rate of 1.2mm.

The CFD model representing TCT in the experimental water channel consisted of two domains. A cylindrical domain enclosing the turbine had a diameter of 0.70m and length of 0.23m. While the external domain representing the channel had a rectangular shape with dimension of [1.375 * 1 *4.755m] as shown in Fig. 2. Steady state analysis were performed and Shear Stress Transport (SST) turbulence model was used with an automatic wall function model. The SST turbulence model has already been successfully used to simulate turbine performance [16-19] because it can model both the near and far wall region, as well as better prediction of flow separation under adverse pressure gradients. A Rotating Frame of Reference (RFR) model was utilized such that the rectangular domain was in a stationary coordinate system. While governing equations for the cylindrical domain were formed in a rotating frame of reference to simulate the rotation of turbine without the rotation of computational grid. All the simulation were considered converged when the RMS residuals reached to 10^-4.

2.1. Boundary conditions

To evaluate the effect of boundary conditions on the performance prediction of turbine, four sets of boundary conditions that can possibly represent a water channel were utilized. The details of the boundary conditions are provided in Table 1. Rotor of the turbine was assigned a no slip
wall condition. Three interfaces were created on the shared faces to connect the meshes of the inner and outer domain with a General Grid Interface (GGI) connection. The boundary conditions (BC#1) actually matches the experimental setup while the other boundary conditions could be the alternate options. The remaining setup was same in all the four simulated cases. A plug flow velocity condition of 1.05m/s and uniform turbulence of 5% intensity was specified at the inlet.

Table 1 Boundary conditions

| BCs | Channel Outlet | Channel Top | Channel Bottom | Channel Side 1 | Channel Side 2 |
|-----|----------------|-------------|----------------|----------------|----------------|
| 1   | Outlet         | Free slip wall | No slip wall   | No slip wall   | Symmetry       |
| 2   | Opening        | No slip wall  | No slip wall   | No slip wall   | No slip wall   |
| 3   | opening        | Free slip wall | No slip wall   | No slip wall   | No slip wall   |
| 4   | opening        | Free slip wall | No slip wall   | No slip wall   | Symmetry       |

3. Model validation
The simulated performance results for BC#1 were compared with experimental data available in [9] over Tip Speed Ratios (TSR) from 2-9 as shown in Fig.3. The performance of the turbine is represented through a performance curve between tip speed ratio and power co-efficient. From Fig. 3 it follows that at lower TSR the CFD over predicts the performance but at higher TSR when the blockage effects are higher CFD under predict the performance than the experiment. However, at the design TSR (5.11) where the turbine produces the maximum power, there is almost 2.8% error between experimental and CFD results.

The simulated results are in acceptable range with the experiment so the accuracy of the CFD model is acceptable. The value of $C_P$ increases with an increase in tip speed ratio and reached to its peak value at TSR 5.1. The same peak value was also observed in the experiment near TSR 5. After passing through TSR 5.1, the value of $C_P$ starts decreasing with further increase in the TSR

![Figure 1. Mesh for CFD simulation](image1)

![Figure 2. Circulating water channel](image2)

4. Results and discussion
4.1. Effect of boundary conditions on the performance of the turbine

To simulate the effect of boundary conditions on the performance of the turbine four different set of boundary conditions were assigned to the channel as described in Table 1. For every set of boundary condition, three simulations at TSRs 2.23, 5.1 and 9.07 were modelled. Results of these simulations are provided in Fig. 4.

As shown in the Fig. 4, the different boundary conditions have a very small effect at lower and higher TSRs, but at optimum TSR where turbine produces the maximum power, the difference in $C_p$ is clearly visible. BC# 1 over predicts $C_p$ at TSR 2.23 by 6% and at TSR 5.11 by 2% compared to the experiment. At TSR 9.07 the $C_p$ is under predicted by 7% than the experiment. BC# 2, 3 and 4 over predict the $C_p$ at TSR 2.23 by 9%, 4% and 7% respectively and at TSR 5.11 by 4%, 6% and 2% respectively. But at higher TSRs where blockage effect and the rotational speed of the turbine were high, $C_p$ was under predicted by 7%, 8% and 6% respectively. At optimum TSR, BC# 3 produces maximum power while the BC#1 that matches to experimental setup produces power closest to the experiment.

To visualize the flow detail around the blade surface, pressure contours and velocity streamlines at the simulated TSRs at the tip of the blade are plotted in Fig. 5. At lower TSR (i.e., TSR 2.23) the flow separation can easily be visualized for every boundary condition throughout the blade surface. The flow separation changes as angle of attack decreases when move from root of the blade towards the tip. As the TSRs increases the flow separation gets weaker and weaker until reached the maximum TSR where the flow is fully attached to the blade surface.
5. Conclusions
This research work reports on the development of a RANS CFD model using Ansys CFX for the performance evaluation of a TCT in an experimental channel. The performance of the turbine was determined based on power coefficient and compared with the experimental results. Results of the CFD simulations were in close agreement to the experimental results. At lower TSRs the developed CFD model over predicted the performance than experiment while at higher TSRs, CFD under predict the performance due to higher blockage effects. Effect of different sets of boundary conditions that could possibly represent an experimental water channel on the performance prediction of the CFD model were analyzed. Changing the boundary conditions did not show much difference in the performance at lower and higher TSRs while at design TSR where the turbine produces the maximum power the difference can be visualized.

6. References
[1] Clarke, J.; Connor, G.; Grant, A.; Johnstone, C. Design and testing of a contra-rotating tidal current turbine. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power* 2007, 221, 171-179.

[2] Coiro, D.; Maisto, U.; Scherillo, F.; Melone, S.; Grasso, F. Horizontal axis tidal current turbine: numerical and experimental investigations, In Proceedings of the Proceeding of Offshore wind and other marine renewable energies in Mediterranean and European seas, European seminar, Rome, Italy, 2006. Citeseer.

[3] Badshah, M.; Badshah, S.; Khalil, S.J.J.O.E.; SCIENCES, A. Hydrodynamic design of tidal current turbine and the effect of solidity on performance. 2017, 36.
[4] Batten, W.; Bahaj, A.; Molland, A.; Chaplin, J.J.R.e. The prediction of the hydrodynamic performance of marine current turbines. 2008, 33, 1085-1096.

[5] Kinnas, S.A.; Xu, W. Analysis of tidal turbines with various numerical methods, In Proceedings of the 1st annual MREC technical conference, 2009.

[6] Mason-Jones, A.; O'Doherty, T.; O'Doherty, D.M.; Evans, P.; Wooldridge, C. Characterisation of a HATT using CFD and ADCP site data. 2008.

[7] Badshah, S.; Badshah, M. Performance prediction of tidal current turbine using coupled fluid structure interaction modelling. 2018.

[8] Badshah, M.; Badshah, S.; Altuf, M.; Jan, S.; Amjad, M.; Anjum, N.A.J.T.J. Research Progress in Tidal Energy Technology-A Review. 2017, 22.

[9] Rahman, N.; Badshah, S.; Rafai, A.; Badshah, M.J.J.o.S.; Research, E. Literature review of ocean current turbine. 2014, 5, 11.

[10] Hafeez, N.; Badshah, S.; Badshah, M.; Khalil, S.J.J.M.S.; Technology, O. Effect of velocity shear on the performance and structural response of a small-scale horizontal axis tidal turbine. 2019, 1-8.

[11] Badshah, M.; VanZwieten, J.; Badshah, S.; Jan, S.I.I.R.P.G. CFD study of blockage ratio and boundary proximity effects on the performance of a tidal turbine. 2019, 13, 744-749.

[12] Jan, S.; Badshah, S.; Amjad, M.; Ahmad, S. Wake Modeling of Tidal Current Turbine Array, In Proceedings of the 2019 International Conference on Power Generation Systems and Renewable Energy Technologies (PGSRET), 2019. IEEE: pp 1-5.

[13] Badshah, M.; Badshah, S.; VanZwieten, J.; Jan, S.; Amir, M.; Malik, S.A.J.E. Coupled Fluid-Structure Interaction Modelling of Loads Variation and Fatigue Life of a Full-Scale Tidal Turbine under the Effect of Velocity Profile. 2019, 12, 2217.

[14] Badshah, M.; Badshah, S.; Kadir, K.J.E. Fluid Structure Interaction Modelling of Tidal Turbine Performance and Structural Loads in a Velocity Shear Environment. 2018, 11, 1837.

[15] Hill, C.; Neary, V.S.; Gunawan, B.; Guala, M.; Sotiropoulos, F.J.S.N.L., Albuquerque, NM. US Department of Energy Reference Model Program RM1: Experimental Results. 2014.

[16] Nobile, R.; Vahdati, M.; Barlow, J.; Mewburn-Crook, A. Dynamic stall for a vertical axis wind turbine in a two-dimensional study, In Proceedings of the World Renewable Energy Congress-Sweden; 8-13 May; 2011; Linköping; Sweden, 2011. Linköping University Electronic Press: pp 4225-4232.

[17] Castelli, M.R.; Ardizzon, G.; Battisti, L.; Benini, E.; Pavesi, G. Modeling strategy and numerical validation for a Darrieus vertical axis micro-wind turbine, In Proceedings of the ASME 2010 international mechanical engineering congress and exposition, 2010. Citeseer: pp 409-418.

[18] Dai, Y.; Lam, W.J.P.o.t.I.o.C.E.-E. Numerical study of straight-bladed Darrieus-type tidal turbine. 2009, 162, 67-76.

[19] Marsh, P.; Ranmuthugala, D.; Penesis, I.; Thomas, G. Performance predictions of a straight-bladed vertical axis turbine using double-multiple streamtube and computational fluid dynamics models. Journal of Ocean Technology 2013, 8, 87-103.

Acknowledgement
The authors gratefully acknowledge the support of Higher Education Commission (HEC) Pakistan for its financial support. The authors also acknowledge the support of Department of Aeronautics and Astronautics at the Institute of Space Technology Islamabad for providing computational facilities at the modelling and simulation lab.