Three Dimensional Numerical Simulations of a Straight-Bladed Vertical Axis Tidal Turbine

P. Marsh\(^1\), D. Ranmuthugala\(^2\), I. Penesis\(^1\) and G. Thomas\(^1\)

\(^1\)National Centre for Maritime Engineering and Hydrodynamics
Australian Maritime College, University of Tasmania, Launceston, 7250, Australia
\(^2\)National Centre for Ports and Shipping
Australian Maritime College, University of Tasmania, Launceston, 7250, Australia

Abstract

The generation of power from our oceans is moving to the forefront in our quest for clean and sustainable energy, and for tidal power vertical axis turbine designs are eminently suitable. Research into these turbines however has lagged behind that of other designs, and further research into their hydrodynamic flow and loading characteristics through the use of Computational Fluid Dynamics (CFD) and Experimental Fluid Dynamics (EFD) is needed to improve their efficiency and longevity. Previous CFD tidal turbine simulations have used Two-Dimensional (2D) simulation models; this work extends these simulations using Three-Dimensions (3D) to fully capture turbine flow characteristics, loading and performance parameters. These transient simulations were performed using a commercial Reynolds Averaged Navier-Stokes (RANS) solver. Theoretical momentum based models have also been developed for comparison. At low tip speed ratio (\( \lambda \)) both CFD and momentum models predicted turbine power accurately when compared to experimental results, but as \( \lambda \) increases CFD models without struts and momentum simulation models showed increasing simulation error due largely to strut drag effects that were not accounted for. The highest level of prediction accuracy across all \( \lambda \) ranges was determined by the use of a 3D CFD model with struts due to the inclusion of the resultant strut drag effects.

Nomenclature

- \( h \): Span [m]
- \( r \): Radius [m]
- \( V \): Inflow Velocity [m/s]
- \( \omega \): Angular Velocity [rad/s]
- \( \rho \): Density of Fluid [kg/m\(^3\)]

Introduction

The need for alternative sources of energy to satisfy our growing energy demands is increasingly apparent, one solution is the utilisation of the tidal and current forces of our oceans. Vertical axis turbines are well suited for generating ocean power as they work independently of water flow direction, making them an excellent choice for tidal power generation in particular. Further research into their hydrodynamic characteristics is necessary due to our limited knowledge of their complex flow properties, as they exhibit high levels of dynamic stall and also complex wake interaction and strut effects. In order to maximise turbine life in the marine environment detailed turbine blade loading simulations are also required, which are dependent upon the accurate determination of the hydrodynamics forces on the turbines as they operate.

Previous CFD work on vertical axis turbines has mainly concentrated upon 2D CFD modelling, due to the considerable simulation time and computing resources required by 3D simulation models. Dai and Lam [2] investigated the performance of a three bladed tidal turbine using 2D CFD analysis methods, and compared results against experimental data at a single \( \lambda \) point. Utilising the k-\( \omega \) Shear Stress Transport (SST) model and a rotating grid with a General Grid Interface (GGI) interface they evaluated performance parameters including torque and power. Lain [3] performed CFD simulations using similar modelling techniques and also favorably compared results with Dai and Lam’s experimental results, again at a single \( \lambda \). Nabavi [4] also used 2D CFD methods to simulate turbine performance, determining reasonable agreement for one experimental test case performed by Rawlings [6] that exhibited low strut drag. However two other experimental cases conducted by Rawlings that exhibited reduced turbine efficiency were not simulated by Nabavi, as 2D CFD models are unable to capture performance losses caused by strut drag.

Methods

This work reports on the ongoing development of 3D CFD and DMS simulations of a straight bladed vertical axis tidal turbine. Simulations using 3D CFD models were performed with and without turbine struts, hubs and shafts to determine their effect on turbine flow characteristics and performance parameters. All simulations were compared to experimental results over a range of operational \( \lambda \) values. Performance parameters including power and efficiency were determined and validated by comparison with experimental results from testing conducted at the Australian Maritime College (AMC) Circulating Water Channel at Beauty Point, Tasmania, Australia, as seen in figure 1. A key parameter of interest is the turbine power coefficient \( C_p \) where

\[
C_p = \frac{\text{Power}}{\frac{1}{2} \rho V^2 S} \tag{1}
\]

and frontal area \( S=2rH \).
All results are compared across a range of $\lambda$, where

$$\lambda = \frac{\rho_0 \omega V}{r}$$  \(2\)

**Experimental Testing**

Testing was performed on a straight three bladed vertical axis turbine in the AMC Circulating Water Channel in 2012. The main turbine dimensions are outlined in Table 1 and the experimental setup with generator, mounting arms and carriage detail is shown in Figure 1.

<table>
<thead>
<tr>
<th>Main Dimensions</th>
<th>Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Blades</td>
<td>3</td>
</tr>
<tr>
<td>Blade Airfoil</td>
<td>NACA634221</td>
</tr>
<tr>
<td>Strut Airfoil</td>
<td>NACA0021</td>
</tr>
<tr>
<td>Radius</td>
<td>0.482 m</td>
</tr>
<tr>
<td>Span</td>
<td>0.482 m</td>
</tr>
<tr>
<td>Blade and Strut Chord</td>
<td>0.07 m</td>
</tr>
<tr>
<td>Number of Struts</td>
<td>6</td>
</tr>
<tr>
<td>Strut Location</td>
<td>1/4 Span</td>
</tr>
</tbody>
</table>

Table 1: Turbine geometry

**Numerical Simulation Methods**

Two methods were used to simulate operational turbine performance characteristics, transient 3D CFD models and Double Multiple Streamtube (DMS) models.

**Momentum Model**

To analyse turbine performance a momentum based mathematical model based on the DMS method outlined by Paraschivoiu [5] was developed. This model is an evolution of the single and multiple streamtube models of Templin [9] and Strickland [8], and uses multiple streamtubes that are split into upwind and downwind parts to calculate the induced velocities using a double iteration process over each individual streamtube [5]. Once the induction factors are known local velocity, angle of attack, normal and tangential forces; and thus torque can be calculated using tables of airfoil section data. As NACA 634221 airfoil data is not available NACA0025 data [7] was used to approximate airfoil lift and drag coefficients. No dynamic stall or strut correction factors were included for simplicity, however correction factors for aspect ratio were included.

**CFD Simulation Methods**

Three dimensional CFD simulations were performed using the commercial ANSYS CFX version 13.0 package, which uses an element-based finite volume method to solve the RANS Equations [1]. For both CFD models with and without struts the same domain sizing, turbulence model and discretisation schemes were used, as outlined below.

**Mesh Generation and Domain Sizing**

Two CFD models were generated; one with all turbine struts, hubs, and shafts, and one without struts, hubs and shafts, to investigate their effects on turbine performance predictions. Unstructured tetrahedral meshes were generated using ANSYS CFX using the boundary conditions outlined in Table 2 for both CFD models. The computational domain used by both models including wake refinement is shown in Figure 2a, with an example of the rotating domain and the inflation layer mesh shown in Figures 2b and 2c. Due to the low blockage ratio of the turbine as tested free stream boundary conditions were used. As no free surface effects were modelled symmetry planes were utilised, thereby reducing mesh size by half. In order to simulate the rotation of the turbine an inner rotating domain was utilised with the interface placed between the stationary outer domain and the rotating inner domain, simulated as a transient rotor/stator using a GGI method. Turbine forces were calculated using ANSYS CFX functions for torque and force, in which the pressure and shear stress distributions around the foils was determined.

<table>
<thead>
<tr>
<th>Item</th>
<th>Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Uniform Flow - 1 m/s</td>
</tr>
<tr>
<td>Outlet</td>
<td>Outlet - Reference Pressure 0 Pa</td>
</tr>
<tr>
<td>Walls</td>
<td>Free slip Wall</td>
</tr>
<tr>
<td>Blades</td>
<td>No slip Wall</td>
</tr>
<tr>
<td>Struts and Shaft</td>
<td>No slip Wall</td>
</tr>
<tr>
<td>Symmetry</td>
<td>Symmetry Wall</td>
</tr>
</tbody>
</table>

Table 2: CFD boundary conditions

![Figure 2: 3D CFD mesh details](image)

(a) Example of 3D mesh domain

(b) Inner rotating domain showing foil and strut layout

(c) Example of inflation layer as used on foils, struts, hubs and shaft

**Turbulence Model and Discretisation Scheme**

For turbulence modelling the k-\omega SST model was used as it has demonstrated high simulation accuracy for flows with high levels of flow separation, dynamic stall and adverse pressure gradient conditions as occur during normal vertical axis turbine operation [3]. A high resolution scheme was used for advection modelling and a second order backward Euler scheme was used to model transient terms. Inlet turbulence setting of 5% was applied to account for turbulent inflow conditions. Residual convergence criteria of $10^{-4}$ with a maximum of 6 internal coefficient loops per timestep was set to ensure convergence. All simulations were run for at least 15 revolutions to remove any initial startup transients and fully develop turbine wake. In order to accurately capture transient results the turbine was rotated 3.6 degrees for each time step, as determined by a temporal independence study. This time step setting was calculated in degrees of rotation per time step to ensure that results were independent of $\lambda$. 

Grid and Temporal Independence

Grid independence studies of domain length, width, height, y+, and time step were conducted for all CFD simulations to ensure results were independent of grid and meshing factors. Overall CFD domain dimensions are defined in figure 3. Mesh size independence was determined for both models at a mesh sizing of 4.2 million and 9.5 million elements without and with struts respectively.

Figure 3: Overall domain sizing dimensions in turbine diameters (D)

Results and Discussion

In order to validate CFD and DMS models comparisons were made between all numerical simulations and AMC Circulating Water Channel testing results at a flow speed of 1 m/s.

Comparison of Results

Experimental and simulated results for $C_p$ are compared in figure 4 over the operational turbine $\lambda$ range.

Figure 4: Comparison of power coefficient ($C_p$) for CFD, DMS and experimental results

Discussion

All simulation $C_p$ predictions show good correlation with experimental testing at low $\lambda$ as seen in figure 4. However at higher $\lambda$ values the models differ greatly in $C_p$ prediction accuracy, due largely to the effects of strut drag on turbine efficiency, which generates resistive torque and thus reduces turbine performance.

Viewing figure 4 the 3D CFD model without struts has predicted turbine performance accurately for low $\lambda$, but shows increasing $C_p$ estimation error as $\lambda$ increases. Significantly when comparing simulated and experimental maximum $C_p$ an error of more than 300% has been determined. Without struts the resistive torque caused by strut drag cannot be simulated, resulting in large $C_p$ prediction error. These errors will also occur for 2D CFD models, as the strut and blade layout of a vertical axis turbine cannot be modelled in 2D.

The full 3D CFD model with struts, hubs and shafts has predicted the performance of the turbine across the $\lambda$ range well when compared to other methods, with the $C_p$ performance curve shifted moderately to higher $\lambda$ values. There is very good agreement at low $\lambda$, but the model overpredicts $C_p$ as $\lambda$ increases. Significantly however this model predicts both the $C_p$ trend and maximum value with a higher degree of accuracy than the other simulation models. This is due to the inclusion of turbine struts which reduce $C_p$ significantly as $\lambda$ increases, as the level of strut drag is related to $\lambda$. This relationship between $\lambda$ and strut drag was determined by Parachiviou [5], who found that secondary effects such as strut drag and shaft effects have greater influence on $C_p$ as $\lambda$ increases. Dai and Lam [2] also determined that turbine $C_p$ was overpredicted at high $\lambda$ ranges when comparing their 2D CFD models with experimental results, as they were unable to account for strut drag using 2D CFD modelling methods.

The DMS model has predicted turbine $C_p$ with reasonable accuracy when compared with experimental testing results as seen in figure 4. At low $\lambda$ ranges $C_p$ is predicted accurately, but as $\lambda$ increases there is a significant reduction in simulation accuracy, as secondary hydrodynamic effects caused by the struts, hubs, and shaft were not accounted for. The accuracy of the DMS model at low $\lambda$ is due to the incorporation of correction factors for blade aspect ratio and tip losses, as well as factors to account for the reduced flow velocity in the downwind turbine area. However as $\lambda$ increases the DMS model overpredicts $C_p$, as it lacks correction factors for strut drag.

Figure 5: Full 3D CFD simulation vortex visualisation - helicity of 6 s$^{-2}$: $\lambda=2$

Figure 6: Full 3D CFD model velocity contours showing wake of struts, shaft, hub and blades: $\lambda=2$
The complex hydrodynamic flow has been simulated by the 3D CFD model with struts, hubs and shafts, and flow interactions and vortices are shown in figures 5, 6 and 7. The reduction in flow velocity caused by the shaft, hub, and strut wake can be seen when comparing figures 7 and 8 for 3D CFD simulations with and without struts, hubs, and shafts. This reduces flow velocity on the blades as they rotate through the downstream turbine area, resulting in the significant decrease of normal and tangential force coefficients, which in turn decreases the total torque. These effects can only be simulated using a complete 3D CFD model, or by adding correction factors to DMS models.

Further research is planned on the full 3D CFD simulation model, and investigations into the effects of turbulence models and surface roughness will be performed. Research will also be carried out into the influence of strut design on turbine performance parameters including $C_p$, in which the strut section drag coefficient will be minimised to reduce overall turbine power losses.

The DMS model $C_p$ predictions were validated successfully at low $\lambda$ ranges. Accuracy at higher $\lambda$ ranges will be improved through the incorporation of corrections factors for secondary effects such as strut drag.

This work is ongoing and will form the basis of the hydrodynamic analysis of a full 3D Fluid Structure Interaction (FSI) study of a vertical axis tidal turbine to investigate structural loading characteristics. Further work to improve and validate results for all operational $\lambda$ ranges will be conducted to ensure the accuracy of the hydrodynamic model used for FSI studies.

Acknowledgments

The authors would like to thank Christopher Hawthorne, Matthew Skledar, Rowan Frost and Alan Faulkner for conducting and providing the experimental testing data.

References


