Numerical Simulation Of A Three Blade Marine Current Turbine

Xin Bai
Eldad J. Avital
Ante Munjiza
John J.R. Williams

Follow this and additional works at: http://academicworks.cuny.edu/cc_conf_hic

Part of the Water Resource Management Commons

Recommended Citation
Bai, Xin; Avital, Eldad J.; Munjiza, Ante; and Williams, John J.R., "Numerical Simulation Of A Three Blade Marine Current Turbine" (2014). CUNY Academic Works.
http://academicworks.cuny.edu/cc_conf_hic/453
NUMERICAL SIMULATION OF A THREE BLADE MARINE CURRENT TURBINE

X. BAI, E. J. AVITAL, A. MUNJIZA, J.J.R. WILLIAMS
School of Engineering and Materials Science, Queen Mary University of London
Mile End Road, London, United Kingdom, E1 4NS

The marine current turbine (MCT) is an exciting proposition for the extraction of renewable tidal and marine current power. However, the numerical prediction of the performance of the MCT is difficult due to its complex geometry, the surrounding turbulent flow and the free surface. In this paper, the authors have developed a 3D Large Eddy Simulation (LES) numerical code to simulate a three blade MCT under a variety of operating conditions based on the Immersed Boundary Method (IBM) and the Conservative Level Set Method (CLS). The results from the current simulations have shown a great match with the published data and provide a promising potential for more extensive study. Furthermore, the deformation of the MCT blades under operating conditions was modeled by coupling the present code with another in-house Discrete Element Method (DEM) code and clear blade deformation was observed which can be used in future to determine the potential life of a MCT.

1 BACKGROUND

Compared with other forms of the renewable energy sources (solar and wind energy etc), tidal power has the distinct advantage of being highly predictable and has an important potential for future electricity generation. The marine current turbine (MCT) is an exciting proposition for the extraction of renewable tidal and marine current power. It is gaining momentum as a viable technology and is currently the subject of much attention and research. Research surrounding MCTs is focused in several areas: power generation, environmental effects, and turbine array design etc. Among these areas, power generation is of the highest priority and attracts the most interest. Therefore, the ability to predict the hydrodynamic performance of a marine current turbine has become essential for the design of the turbine and various numerical and experimental methods have been proposed to achieve that. In addition, the structural load, material fatigue and the free surface wave influence etc have also received a substantial research interest as their impact on the power production and expected life of the turbine.

Among all the research surrounding the MCTs, the experimental study is the most straightforward and informative way to investigate their performance. Bahaj et al. [1] have performed extensive experimental analysis of the performance of a horizontal axis marine turbine in a towing tank and cavitation tunnel. Results included identification of 20° as the hub pitch angle producing the maximum power coefficient. However, as the experimental study can be both
time-consuming and expensive, numerical simulations are more preferable for testing large scale or a number of different designs. Kinnas and Xu [2] have applied the boundary element method coupled with a potential flow solver to predict the wake geometry and cavity pattern behind a marine current turbine. Another numerical method based on blade element momentum theory (BMET) has also been successfully developed to give a rough prediction of turbine performance but cannot provide detailed information about the surrounding fluid field [3]. Much of current marine turbine research mentioned above are informed by previous work on wind turbines and has taken no consideration of the real operating condition of the MCTs, such as the effect of the free surface, turbulence in surrounding flow and wave induced velocity. In the most recent work, Computational Fluid Dynamic (CFD) simulations have been conducted to handle these problems. For example, significant work have been carried out by Consul et al. [4] on the hydrodynamic performance of a cross-flow marine turbine under difference blockage and free surface conditions using FLUENT and the RANS approach.

In this paper, the simulations of a three-blade horizontal axis MCT have been carried out for a series of operating conditions. The power and thrust coefficient has been calculated and compared against experimental data. The structural load and blade deformation were also illustrated for the consideration of material fatigue.

2 METHODOLOGY

The 3D in-house CFD code CgLes [5] was used for solving the fluid field in all the simulations. This code has been used for many years by several researchers on UK national high-end computing facilities and is highly parallelized and efficient. The fluid-structure interactions were captured using the direct forcing Immersed Boundary Method, which was first developed by Mohd-Yusof [6] to handle the flow around a rigid body. In this method, the immersed boundary was represented by a set of discrete points of which the physical velocity are already known. The forces are calculated from the difference between the virtual velocity of point interpolated from the nearby fluid grids and its known velocity and then exerted back to the fluid grids to correct the flow field. This direct-forcing method was later successfully used to simulate flow over 3D complex geometry [7] and moving boundaries coupled with a turbulence model [8]. In the current simulations, the original direct-forcing IBM scheme was improved by using a novel iterative body force distribution scheme, where the body force updating is incorporated into the pressure iterations so that the boundary condition on the immersed boundary can be satisfied exactly. More details of the IBM scheme and its implementation in CgLess can be found in the work of Ji et al. [9].

2.1 Governing Equations

For turbulence modelling, the spatial filtering based LES was chosen for its ability of capturing the unsteadiness of relatively small turbulence while requires less computational resources than Direct Numerical Simulation (DNS). Let “~” denotes the filtered value of one variable, the momentum equation can then be written as:

$$\frac{\partial \bar{u}_j}{\partial t} \Delta V + \int_{s} \bar{u}_j \bar{u}_i \rho \frac{\partial }{\partial x_i} \Delta V + \int_{s} \left( \frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j} \right) \nu \Delta V$$

Where \( u \) is the velocity, \( \rho \) is the pressure, \( \nu \) is the kinematic viscosity and \( f \) is the external force, \( \tau_{ij}^{\sigma} \) is the sub-grid stress tensor used to take account of the effect of unsolved length scales and the correct approximation of \( \tau_{ij}^{\sigma} \) should account for the reminder of the energy spectrum and
the interaction among all length scales. In the current simulations, the sub-grid tensor are approximated according to Boussinesq eddy viscosity assumption and modelled using the Mixed-Time-Scale (MTS) model [10].

2.2 Conservative Level Set Method

A modified conservative Level Set Method developed by Olsson and Kreiss [11] is employed to handle the free surface effect on top of the operating turbine. Unlike the traditional level set method, the level set function in the current CLS method was assigned to represent the phase field instead of the distance field. To keep density and viscosity discontinuities over the interface, the Heaviside function was employed.

\[
H(\varphi) = \begin{cases} 
1 & \varphi > 0 \\
0.5 & \varphi = 0 \\
0 & \varphi < 0
\end{cases}
\]

Where \( \varphi \) is the signed distance in traditional level set method. To avoid the numerical instabilities introduced by the sharp jump of level set function, a hyperbolic tangent function was used to smear out the interface.

\[
H(\varphi) = \frac{1}{2} \left( 1 + \tanh(\varphi / 2\varepsilon) \right)
\]

where \( \varepsilon \) denotes the spreading width of \( H \). Then the material properties like density and viscosity can be assigned to each phase using their level set values. Once the values of level set function have been constructed, it can be advected in a conservative way. To maintain the shape and width of the interface after the advection, an re-initialization step was introduced by adding a conservative compression term and a small viscosity term. Let \( H \) denotes the level set function, \( n \) denotes the normal vector of the interface, \( \tau \) is the artificial time for re-initialization, and the equations for advection and re-initialization are shown as follows.

\[
\frac{\partial H}{\partial t} + \nabla \cdot (uH) = 0
\]

\[
\frac{\partial H}{\partial \tau} + \nabla \cdot \left( H \left( 1 - H \right) \hat{n} \right) = \nabla \cdot \left( \varepsilon \left( \nabla H \cdot \hat{n} \right) \hat{n} \right)
\]

The CLS method mentioned above has a conservative feature and has proved to be capable of handling problems with large density ratios (about 1000 to 1), surface tension and viscosity jump conditions with a high order of accuracy. In advection of Eq. (5), the total variation diminishing (TVD) scheme was used to enforce the conservation law without introducing oscillations near the interface. The finite volume approach to calculate the flux in each control volume and reconstruct the interface using a piecewise linear scheme. Detailed information about the current scheme and its practical applications can be found in the previous work of the authors [12].

2.3 Coupling of the Fluid and Solids
As mentioned above, the interactions between the turbine and surrounding fluid can be simulated using the IBM scheme. In real operating conditions, the forces from the surrounding flow also serve as the structural load on the turbine and will lead to blade deformation. Such deformation may result in the reduction in power production over time and finally leads to turbine failure. To take that into account, the combined finite-discrete element method (FEM–DEM) was used to simulate the deformation of the turbine blades under the various forces developed in the fluid [13]. The in-house cutting-edge Y-code comprises a set of C libraries incorporating the latest breakthroughs in discontinua simulations. It is capable of modelling the movement, deformation, fracture and collision of millions of solid bodies of different shapes and sizes.

In order to capture the blade deformation, an independent set of mesh was created for the solid body of the MCT. The surface nodes of the solids were treated as immersed boundary points in the fluid to form the correct boundary condition. The fluid force acting on the boundary nodes was interpolated from the grid points and then feed to the Y-code to calculate the deformation of the blades and update the coordinates of the IB points, which can be used to solve the flow field in the next time step. The coupling of the CgLes and Y-code has greatly expanded the research scope of the numerical simulations and has been successfully applied in a number of simulations, for example, sediment transport in turbulent flow [14], red blood cells [15] and flow in a ureter muscle [16].

3 VALIDATIONS AND RESULTS

Extensive validation cases of the IBM and CLS schemes as well as the turbulence models used in the current simulations can be found in the authors’ previous work [9] [14] [17] [18] including flow over static and oscillating cylinder, standing wave, break dam problem and etc. For the sake of simplicity, the results from two cases were presented in this paper to show the accuracy and reliability of the current numerical scheme: a) turbulent flow past a static turbine; b) turbulent flow past a rotating turbine.

3.1 Turbulent flow past a static turbine

The turbine design of Bahaj et al. [1] is chosen for this project’s validation purposes as large associated data-set are available. The selected model turbine has a diameter of 800mm and consisted of three blades developed from the NACA 63-8XX series airfoil sections. There are 17 stations along each blade of which the coordinates are interpolated from coordinate-based data for NACA 63-812, 63-815, 63-818, 63-821 and 63-824. These simple profiles are given different thickness characteristics, chord length, and pitch distributions to form a complicated 3D twisting blade- see Figure 1.

Figure 1. The 3D Geometry of the blade
The normalized dimensions of the computational box using the rotor diameter are set as [-2, 7] × [-2, 2] × [-2, 2] in the x, y, z direction respectively. The computational domain was discretized on a stretched Cartesian mesh with a grid resolution of 512×320×320. The grids were stretched away from the solid while in the vicinity of the turbine, uniform rectangle grids were set to resolve the flow field with the finest grid spacing of 1/256. The average y+ near the turbine is about 30. The boundary condition is selected to with inflow and outflow conditions in the streamwise direction, free-slip wall in the spanwise direction and a free surface approximation on the top wall. The free stream inlet velocity is set as 1.0 m/s, so the Reynolds number based on the rotor diameter can be worked out to be 8x10⁶.

Figure 2 shows the time averaged pressure contour over the static marine current turbine in the current simulation. It can be seen that the pressure distribution along each blade is almost the same, with positive pressure region at the front of the blade and negative pressure at the back of the blade which is in consistent with the expected distribution for flow directly onto a bluff body. On the front of the blade, the highest pressure region presents at the mid-span of the blade facing the inflow. On the back of the blade, the negative pressure region is more dominant and covers the whole surface – see Figure 3. Among the negative pressure regions, the largest negative pressure value is found to be near the blade tip. The gradient between the high and low pressure regions at two faces indicate a strong force acting on the blade which would aid the rotation of the device in a rotating circumstance. In the current case, looking from the direction of inflow, the force generated by the interaction of the blade and surrounding flow will drive the turbine to rotate in clock-wise direction. Once rotating, the low pressure region behind the blade is expected to adjust in a clockwise direction to give the lift characteristics of the profiles and again aid the rotation.
3.2 Turbulent flow past a rotating turbine

The hydrodynamic performance of the full scaled marine turbine was investigated in the full scaled computational domain, the normalized dimensions of the computational box using the rotor diameter are set as [-3, 7] [-1.95, 1.05] [-2.5, 2.5] in the x, y, z direction respectively. The computational domain was then discretized on a stretched Cartesian mesh with a grid resolution of 640x320x512. For the turbine body, there are 262,729 discrete Immersed Boundary Points. The free stream velocity is set as 1.73 m/s as used by Bahaj et al. [1]. The Reynolds number based on the rotor diameter is worked out to be 1.4x10^6.

In Figure 4, the power and thrust coefficient at three different TSR speeds were plotted and compared with the experimental data. For the turbine design with hub pitch of 20°, it can be seen that the current result agrees well with the published data and catches the variation of the power coefficient correctly. As the rotation speed increases, the power coefficient first increases and then decreases after reaching its maximum. This can be used to determine the optimized operation speed of the turbine if installed in a location with certain range of flow speed. The maximum power coefficient of 0.45 was observed at TSR = 6 which is only 0.5% smaller compared with the experimental data. The small deviation (about 2%) at TSR = 8 can be explained by low resolution of the blade tip. As the section profile at the blade tip has a chord length of only 0.025xDc, there is only 10-13 cells along the chord and even fewer along the thickness directions. As the rotational rate increases, the power generated by the tip area may not be captured correctly and the overall power coefficient will be smaller than expected.

For the thrust coefficient, $C_T$ was found out to increase with the increase of rotational speed. At TSR = 6, the thrust coefficient was found to be 0.80 which is the same as the experimental data. The detailed torque and thrust value of each blade at TSR = 6 was presented in Table 1. The torque and thrust value for every blade was found out to be almost the same, which means the simulation is statistically stable.

![Figure 3. Pressure contour on the blade of the static marine current turbine](image)

![Figure 4. Power and Thrust coefficient at different rotating speed against experimental data](image)

3.3 Blade Deformation

From the previous simulations, the performance of the turbine with rigid blades has been captured correctly using the proposed numerical scheme. To simulate the blade deformations, the coupled $CgLes$-Y code was employed to take account of the blade flexibility. In order to
expedite the deformation process, a young’s modulus of 80MPa is selected for the turbine blade, which is about 100 times smaller than the modulus of the standard steel. The Possion ratio is set as 0.3 and the density of the solid body is set as 7000 kg/m$^3$.

The simulation was restarted using the flow field information from previous full scale simulation and continued to run for a further 10,000 rotations at TSR = 6. The deformation of the blade at different stage was shown in Figure 6. In Figure 5(a), the blade is at its initial condition and start to deform after 2,000 rotations as seen in Figure 5(b). The highest stress was observed at the root of the blade close to the trailing edge of each profile. In Figure 5(c), after 5,000 rotations, a clear twist can be seen on the blade and the deformation process also start to accelerate. At the time of 10,000 rotations, the blade has lost its original shape and have bended towards the clockwise direction. In that case, the MCT have deviated from its optimal operating condition and its power production has reduced about 40%.

4 CONCLUSION

The immersed boundary method and free surface method was presented in this paper to aid the simulation of a marine current turbine in turbulent flow. The numerical result shows that the authors’ scheme has great potential in simulating flows with complex geometry and moving boundaries. To handle the blade flexibility problems, Combined Discrete Element Method was employed and coupled with the fluid solver. A clear deformation of the blades has been observed using the coupled code together with reduced power output and provide an insight of the turbine failure at real operating conditions. Such a conclusion is very promising and future work may involve carrying out the simulation of the MCT in extreme conditions and several MCTs working in arrays. Another proposition is to investigate the material fatigue when operating by recording the force variations under different operation conditions.

Figure 5. Power and Thrust coefficient at different rotating speed against experimental data
REFERENCES


