Renewable Energy 63 (2014) 715–723

Contents lists available at ScienceDirect

Renewable Energy

journal homepage: www.elsevier.com/locate/renene

Numerical simulation of a marine current turbine in free surface flow

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 E1 4NS, United Kingdom

ARTICLE INFO

Article history:
Received 4 January 2013
Accepted 25 September 2013
Available online

Keywords:
Marine current turbine
Immersed boundary method
Free surface
Level set method
Power performance
TSR

ABSTRACT

The numerical prediction of the power performance of a marine current turbine under a free surface is difficult to pursue due to its complex geometry, fluid–structural interactions and ever-changing free surface interface. In this paper, an immersed boundary method is used to couple the simulation of turbulent fluid flow with solid using a three-dimensional finite volume solver. Two free surface methods are proposed and tested for different conditions. The methods were then validated respectively by various studies and a coupled simulation was proposed for a marine current turbine operating under free surface waves. The power coefficients of a horizontal axis marine current turbine (MCT) with different rotating speeds are calculated and compared against the experimental data. It is found that the method is in general agreement with published results and provides a promising potential for more extensive study on the MCT and other applications.

1. Introduction

Renewable energy usually refers to those natural sources of energy which are possible to use without diminishing the resource. Modern renewable energy technology can date from the late 20th century and is thought to provide about 9.7% of final energy consumption worldwide in 2011and is still increasing [1]. The current European target is to source 20% of its energy from renewable sources by 2020 and the UK was given a 15% target to aid this.

Among all the renewable energy sources, tidal power has the distinct advantage of being highly predictable compared to some other forms of renewable energy (solar energy, wind energy, etc) which gives tidal energy development an important potential for further electricity generation. The marine current turbine (MCT) is an exciting proposition for the extraction of tidal and marine current power. It is gaining momentum as a viable technology and is currently the subject of much attention and research [2].

Research surrounding MCT's 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. In a report by the European Commission [3], 106 promising locations for marine current exploitation have been identified and given an estimated total electricity output of 48 TWh per year. Based on the UK electrical consumption of 328 TWh for 2010 [4], the MCTs have a potential of supplying more than 10% of the current UK energy demand. Therefore, the ability to predict the hydrodynamic performance of a marine current turbine has became essential for the design of the turbine and various numerical and experimental methods have been proposed to achieve that. Significant work have been carried out by Consul et al. [5] on the hydrodynamic performance of a cross-flow marine turbine under difference blockage and free surface conditions using FLUENT and the RANS approach. Kinnas and Xu [6] 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. Bahaj et al. [7] have performed extensive experimental analysis of the performance of a horizontal axis marine turbine in a towing tank and cavitation tunnel. Another numerical method based on blade element momentum theory (BMET) has also been successfully developed to give a rough prediction of turbine performance [8] but can't provide detailed information about the surrounding fluid field.

In this paper, a numerical scheme based on large eddy simulation and the immersed boundary method is used to capture fluid–structure interaction. It was first proposed by Peskin [9] to simulate the flow around an elastic body and in 1997, a direct forcing IB method was developed by Mohd-Yusof [10] to handle a rigid body. In this method, the immersed boundary was represented by a set of discrete points of which the physical velocity is already known. The force is calculated from the difference between the virtual velocity of point which is interpolated from the nearby fluid grid and its known velocity and added back to produce the correct flow field. This direct-forcing method was later successfully used to simulate flow over 3D complex geometry [2] and moving boundaries coupled with a turbulence model [11]. Recent development of the immersed

* Corresponding author. Tel.: +44 02078825308.
E-mail address: x.bai@qmul.ac.uk (X. Bai).

0960-1481/$ – see front matter
http://dx.doi.org/10.1016/j.renene.2013.09.042

boundary method [12] have made it possible to treat fluid-solid interface in a partitioned approach by exchanging geometric, kinetic and dynamic information between fluid and solid and therefore can be used for large scale and parallelized simulations.

2. Computational method

2.1. Governing equations

To simulate the flow in a 3D computational domain the authors used an in-house Computational Fluid Dynamic (CFD) C code called CgLes [13]. 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 governing Navier–Stokes equations for incompressible flow are written in their finite volume format:

$$\int_V \frac{\partial u_i}{\partial x_i} dV = 0 \tag{1}$$

$$\int_V \frac{\partial u_i}{\partial t} dV + \int_V \frac{\partial u_i u_j}{\partial x_j} dV = -\int_V \frac{1}{\rho}\frac{\partial p}{\partial x_i} dV + \frac{1}{\rho}\int_V \frac{\partial T_{ij}}{\partial x_j} dV + \int_V \frac{f_i}{\rho} dV \tag{2}$$

where ρ is the density, u is the velocity vector, f is a volume forcing term (gravity, feedback force and etc) and i ranges from 1 to 3 to represent three directions and velocity components respectively. T is the component of the total stress tensor which for a Newtonian fluid, can be calculated by

$$T_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\nabla \cdot u_j\right) \tag{3}$$

where μ is the dynamic viscosity of the fluid and δ_{ij} is the Kronecker's delta.

For turbulence modelling, the spatial filtering based Large Eddy Simulation was chosen for its ability of capturing the unsteadiness of relatively small turbulence while requires less computational resources than Direct Numerical Simulation. Let ""denotes the filtered value of one variable, the momentum equation can then be rewritten as:

$$\frac{\partial \overline{u_i}}{\partial t}\Delta V + \int_s \overline{u_i}\overline{u_j} n ds + \int_s \tau_{ij}^{sg} n ds = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i}\Delta V + \int_s \vartheta\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) n ds + \frac{\overline{f_i}}{\rho}\Delta V \tag{4}$$

where τ_{ij}^{sg} is the sub-grid stress tensor used to take account of the effect of unsolved length scales and the correct approximation of τ_{ij}^{sg} should account for the reminder of the energy spectrum and the interaction among all length scales. Using Smagrinsky Sub-grid model with the Manson wall damping function, the sub-grid stress tensor can be approximated by

$$\tau_{ij}^{sg} = -2\vartheta_t \overline{S_{ij}} + \frac{1}{3}\tau_{ii}^{sg}\delta_{ij} \tag{5}$$

$$\vartheta_t = \sqrt{2\overline{S_{ij}}\overline{S_{ij}}} * 1.0 / \left(\frac{1}{(C_s\Delta)^2} + \frac{1}{(\kappa l_w)^2}\right) \tag{6}$$

where $\overline{S_{ij}} = 1/2((\partial\overline{u_i}/\partial x_j) + (\partial\overline{u_j}/\partial x_i))$ is the rate-of-strain tensor, ϑ_t is the turbulent eddy viscosity, Δ is the cube root of the cell volume, C_s is a model constant taken as 0.095, l_w is the distance from the wall boundary and $\kappa = 0.415$ is the Von Karman constant.

At the free surface interface, care has to be taken to avoid a sudden jump in phase properties which will lead to unsteadiness of the solution. The density and viscosities are therefore specified as follows:

$$\rho = \rho_g + \left(\rho_l - \rho_g\right)c \tag{7}$$

$$\mu = \mu_g + \left(\mu_l - \mu_g\right)c \tag{8}$$

ρ_l, ρ_g, μ_l, μ_g represent the density and dynamic viscosity of the water and air respectively. c is the fractional volume scalar which is 1 when the computational cell is full of water and 0 if occupied by air. The volume fraction is updated with the change of the surface profile and can be obtained using either the height function or level set function.

Once all the values of the density and viscosity have been known, the projection method is used to derive the pressure Poisson equation.

$$\nabla \cdot \left(\frac{\nabla p}{\rho}\right) = \frac{\nabla \cdot u^*}{dt}, \tag{9}$$

where u^* is the intermediate velocity vector from the predictor step and dt is the time stepping [13].

Equations (1)–(9) are then discretized on a fixed staggered Cartesian grid and solved by the finite volume approach. A second-order Adams-Bashforth method has been used for time integration, and the following pressure solvers have been used and compared against each other for efficiency, the Conjugate Gradient (CG) method, the dynamic Successive-Over-Relaxation (SOR) method and the Bi-conjugate gradient stabilized (BICGSTAB) method. The detailed result will be given in the case study section.

2.2. Immersed boundary method

The immersed boundary method used in this paper is based on the direct forcing IB method of Mohd-Yusof [10] with an improved body force distribution scheme. Also, in our implementation the body force updating is incorporated into the pressure iterations so that the boundary condition on the immersed boundary can be satisfied exactly [14].

Using the IB method and the second order Adam-Bashforth time marching scheme, in a single phase approximation, the momentum equation can be discretized and rearranged to get

$$u_i^{n+1} = u_i^n + dt\left(\frac{3}{2}h_i^n - \frac{1}{2}h_i^{n-1} - \frac{3}{2}\frac{\partial p^n}{\partial x_i} + \frac{1}{2}\frac{\partial p^{n-1}}{\partial x_i}\right) + f_i^{n+1/2}dt \tag{10}$$

where

$$h_i = -\frac{\partial u_i u_j}{\partial x_j} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[(\mu + \rho\vartheta_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{11}$$

comprises the convective and diffusive terms and the force f is the body force representing the virtual boundary force, the subscript n denotes the current time step and $n-1$, $n+1$ stand for the values from previous step and future step respectively.