Renewable Energy 57 (2013) 269–282

Contents lists available at SciVerse ScienceDirect

Renewable Energy

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

Numerical modelling of the effect of turbines on currents in a tidal channel – Tory Channel, New Zealand

David R. Plew [a,*], Craig L. Stevens [b,1]

[a] National Institute of Water and Atmospheric Research, PO Box 8602, Riccarton, Christchurch 8440, New Zealand
[b] National Institute of Water and Atmospheric Research, Wellington, New Zealand

ARTICLE INFO

Article history:
Received 1 June 2012
Accepted 3 February 2013
Available online 1 March 2013

Keywords:
Tidal energy
Turbines
Tory channel
Cook strait
Turbine drag
Hydrodynamic model

ABSTRACT

Numerical modelling is used to assess the effect of a turbine array on tidal currents in the Tory Channel, New Zealand. The Tory Channel is the smaller of two entrances from Cook Strait to the Queen Charlotte Sound with a large island separating the narrow Tory Channel from the main entrance. The 2D depth-averaged finite element model is validated against velocities from shipboard ADCP transects from a survey during spring tide conditions, and water levels recorded at the study site. Turbine drag is introduced to the model as a stress term in the momentum equations, and includes both the turbine thrust and the structural drag. Turbine array drag is a function of the number and size of turbines, which can be parameterised in a non-dimensional number. This non-dimensional turbine drag number *D* can be used to represent the drag of several different turbine designs. Restrictions are placed on the size of the array to ensure that turbines are placed in realistic locations. In this study, turbines are restricted to areas with water depths greater than 30 m, and where spring tide currents (in the absence of turbines) are greater than 2.0 m s^{-1}, and consequently the turbine array does not span the entire channel width or length. The modelling shows turbines will reduce current speeds both within the turbine array, and also throughout much of the Tory Channel, with local increases in speed immediately adjacent the array. Cut-in and maximum or rated turbine speeds are also incorporated to compare how these factors influence both the power production and effect on currents. The study shows that, due to the restrictions placed on the array location, the likely power production that can be achieved is considerably less than what an analytical prediction suggests might be obtained from the channel. Due to the effects of turbines on current speeds, optimising the area occupied by an array is likely to be an iterative procedure. The power produced per turbine unit could be substantially improved, with little impact on total power produced by the array, by removing turbines from areas where power produced was low. Turbine operational limits, applied in the form of cut-in speeds below which no power is produced, and design speeds above which load shedding occurs, affect both the magnitude and spatial distribution of power production and thus need to be considered in array design.

1. Introduction

Tidal currents appear to be an attractive source of “clean” energy. Tides are predictable, more so than wind, river flow or wave conditions, and it is possible to forecast potential power generation years in advance with a reasonable degree of accuracy. The first step when considering the potential of a site for tidal stream power generation is to assess the available resource through measurements of current speeds [1], or through numerical modelling [2–4]. However, it is recognised that the extraction of energy by, and the drag from, in-stream turbines will affect currents [5–7]. At the time of writing, there are only a few operational, individual, in-stream turbines. However, proposed tidal power systems will use arrays of turbines that could conceivably occupy large areas. Other examples of arrays of structures introduced to the marine environment, such as lines and cages used in aquaculture, affect currents over large areas, and well beyond the immediate bounds of the arrays [8–10]. Similar effects may be expected for tidal turbine arrays. Turbine-induced changes in currents can affect not only the extraction of energy, but may also have ecological implications such as changing transport of sediment [11] and suspended material [12].

Both analytical and numerical approaches have been used to assess the effect of turbine arrays on currents and water levels, and

* Corresponding author.
E-mail addresses: david.plew@niwa.co.nz (D.R. Plew), craig.stevens@niwa.co.nz (C.L. Stevens).
1 Private Bag 14901, Wellington, New Zealand.

0960-1481/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.02.001

to estimate the optimal number of turbines as well as the power that can be extracted. Analytical models can provide a useful first estimate of the likely tidal resource, but they rely on simplified channel geometry and generally a 1D assumption. In the simplest form of analytical model, turbines arrays are evenly spread over the channel. These models demonstrate that increasing the number of turbines will initially increase the total power extracted, until the drag from the turbines slows currents to a degree that additional turbines results in a reduction of total power extracted. The total array drag must be optimised to maximise power output [5,7,13,14]. Other analytical models have considered arrays that span part of a channel width [6,7,13]. Common to all analytical models is that the array drag is evenly distributed over all or part of the channel.

In contrast, numerical hydrodynamic modelling can provide more detailed information particularly where large spatial differences in current speeds occur along or across a channel. To obtain the highest level of detail about flow modification and power production, it is possible to model each individual turbine in 2D or 3D numerical models [15]. This approach requires knowledge of turbine properties and also the exact location of turbines. The computational costs of such a modelling approach are high, particularly if different turbine numbers, locations or designs are to be compared, and it is likely that such detailed modelling would be conducted in the later stages of assessing a tidal energy extraction project. An intermediate approach that can be implemented more readily is to distribute turbine drag over the area covered by a proposed array [11,12,16,17]. This level of numerical modelling, that includes turbine drag, is likely to follow assessments of the tidal energy resource based on kinetic energy flux [2,3,18,19]. The kinetic energy flux approach does not allow for the flow reduction caused by the turbines and so will give inaccurate predictions of potential power production, but may be used to identify regions in which more accurate assessments are desired.

There is some variation in how the effect of turbine drag is included in numerical models. The most common approach is to add a sink or drag term into the momentum equation [4,11,12,15,16]. However, this term has been parameterised in different ways. In some studies the turbine drag term is estimated from the extracted power [4,11], while elsewhere the drag is calculated from the turbine thrust and drag terms [12]. This second approach is more complete in that it allows for inefficiencies in power extraction (not all of the energy removed from the flow goes to power generation, some is lost to turbulence), and may also include the drag from the structures supporting the turbine. While it is convenient to assume that drag and thrust coefficients are constant, in many cases turbines may have minimum or cut-in speeds or maximum design speeds, beyond which turbines either cease operating or power output is limited [20]. Consequently the relationship between drag and velocity may vary over the velocity range in which the turbine is placed.

In this study, numerical modelling is used to investigate the potential of a channel for tidal stream power generation. The modelling is based on the Tory Channel Entrance, located near the northern end of the South Island, New Zealand. The Tory Channel Entrance has attracted the interest of tidal power developers because of its strong currents and proximity to infrastructure.

A unique feature of the Tory Channel is that it is formed by an island, thus there is an alternative entrance for flow into the Queen Charlotte Sound. The goals of this study are to

- predict how increasing numbers of turbines will affect hydrodynamics at the array/channel scale with flow being diverted around the array
- and similarly at the Sound scale, whether flow is instead diverted to the other side of the island forming the channel
- estimate the power that could be extracted by different numbers of turbines, and how this power varies over tidal cycles
- determine if incorporating limits to turbine operation (cut-in and design speeds) substantially alters the effect of the turbine array on hydrodynamics and power production

In the Methods section of this paper, we consider the drag on, and power produced by a single turbine. Further to this, we then examine how cut-in and maximum design speeds affect the drag and power, and how these can be incorporated into a numerical model using a non-dimensional turbine drag parameter that is independent of turbine size. A numerical model and the study site it is applied to are also described. In the Results, numerical model predictions of the flow in the absence of turbines are compared to field data, and then the model is used to investigate how currents and power production are affected over a range of non-dimensional turbine drag, both with and without turbine operating limits. Key findings, limitations of the present approach and improvements are considered in the Discussion.

2. Methods

2.1. Turbine power and drag

With regard to modelling turbines, the two parameters of interest are the power produced by the turbine, and the thrust (drag) produced by the turbine. The power P produced by a turbine may be expressed by the following equation:

$$P = \frac{1}{2} C_P \rho A U^3. \tag{1}$$

The thrust on the turbine T is

$$T = \frac{1}{2} C_T \rho A U^2. \tag{2}$$

A is the cross-sectional area of the turbine (generally taken as the area swept by the blades), U the upstream flow velocity, ρ the density of sea water, C_P a power coefficient, and C_T a thrust coefficient. Published values of the power and thrust coefficient for scale model turbines are often derived from axial force and torque measured at the turbine hub so do not include mechanical losses in generator efficiencies (which reduce the power produced), or drag from the main body (nacelle) and supporting structures [21]. Thus, the total drag F_D from a turbine should include both thrust and structural drag:

$$\begin{aligned} F_D &= \frac{1}{2}\rho C_T A U^2 + \frac{1}{2}\rho C_D A_s U^2 \\ &= \frac{\rho U^2}{2}(C_T A + C_D A_s). \end{aligned} \tag{3}$$

The first term on the right side of the equation above is the thrust, while the second term is the structural drag. C_D is a drag coefficient for the structural components, and A_S the projected area of the structural components. As described previously, in some models the turbine drag is parameterised from the power term as $F'_D = P/U = 1/2 C_P \rho A U^2$. If this method is used, the drag will be underestimated as the structural drag is not included, and we might reasonably expect that the thrust coefficient is greater than the power coefficient (i.e. $C_T > C_P$) otherwise more power would be produced than work done against the flow. This is also supported by data for both horizontal and vertical axis turbines [21,22].

The power and thrust coefficients are influenced by design and operation factors including blade pitch, tip speed ratio and yaw. Some turbine designs may allow the blade pitch angle and yaw to be adjusted, while other designs may use a fixed blade and/or yaw to reduce mechanical complexity and the likely cost of maintenance.