Renewable Energy 63 (2014) 477–485

Contents lists available at ScienceDirect

Renewable Energy

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

Simulations of a vertical axis turbine in a channel☆

Anders Goude*, Olov Ågren

Uppsala University, Ångström Laboratory, Division of Electricity, Box 534, 751 21 Uppsala, Sweden

ARTICLE INFO

Article history:
Received 23 October 2012
Accepted 25 September 2013
Available online

Keywords:
Vertical axis turbine
Vortex method
Channel flow
Simulation
Current power

ABSTRACT

The power coefficient of a turbine increases according to the predictions from streamtube theory for sites with a confined fluid flow. Here, a vertical axis turbine (optimized for free flow) has been simulated by a two-dimensional vortex method, both in a channel and in free flow. The first part of the study concerns the numerical parameters of channel simulations. It is found that for free flow and wide channels, a large number of revolutions is required for convergence (around 100 at the optimal tip speed ratio and increasing with higher tip speed ratio), while for smaller channels, the required number of revolutions decreases.

The second part analyses changes in turbine performance by the channel boundaries. The turbine performance increases when the channel width is decreased, although the results are below the predictions from streamtube theory, and this difference increases with decreasing channel width. It is also observed that the optimal tip speed ratio increases with decreasing channel width. By increasing the chord, which decreases the optimal tip speed ratio, the power coefficient can be increased somewhat.

1. Introduction

If a site for a turbine is bounded by rigid walls, the walls will affect the character of the flow and thereby the performance of the turbine. One example is a river, where the net flow between impermeable river banks is constant. In this case, the flow boundaries increase turbine performance [1,2]. The same holds for experimental tests in wind tunnels and towing tanks etc. Another example is flow generated by tides in a river mouth. Here, the flow resistance caused by the turbines can decrease the total flow through the channel and reduce power production [3]. A final example is flow with artificial walls created in open sea. Here, the flow has the possibility to pass around the walls, and the fluid motion between the channel walls depends on the flow resistance of the turbine. This case has been studied in Refs. [4–6] and elsewhere. The present study only covers the first case, where the total flow in the channel is constant.

Apart from building turbines in confined areas, the study of turbines inside a channel is also important from a simulation point of view. For simulations based on volume discretization, e.g. FEM, walls in the simulation domain are introduced. If these walls are located too close to the turbine, the computed performance will be incorrect. This article is intended to be useful both for designers of turbines in channels and for those involved with simulations of turbines.

In this study, straight bladed vertical axis turbines are investigated. For a simple and robust turbine design, only turbines with a fixed blade pitch angle are studied.

Vertical axis turbines are typically simulated with either streamtube models, vortex models or CFD models based on volume discretization. Since the traditional streamtube models [7,8], only work for free flow, this is not an option for channel flow. The choice therefore stands between vortex methods, which in their basic forms are best suited for free flows, and CFD models, which due to the volume discretization always are accompanied by channel effects. To study the channel effects properly, many simulations have to be performed. Therefore, a computationally efficient method is required, and vortex method simulations are generally faster than CFD models. For this reason, the vortex method is chosen for the current study. To further reduce the computational time, only 2D simulations are performed, which is a reasonably realistic model for a vertical axis turbine.

2. The vortex method

Vortex methods are based on the incompressible Navier–Stokes equations, but instead of using the velocity as the main variable, the

* Corresponding author.
E-mail address: anders.goude@angstrom.uu.se (A. Goude).

0960-1481/$ – see front matter © 2013 The Authors. Published by Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.09.038

vorticity (i.e. the curl of the velocity) is chosen. The vorticity field is discretized in a grid-free manner using point vortices. In the vorticity formulation, the vorticity is a conserved quantity, which can only be created at the boundaries and will move with the flow velocity. Vortex methods are well documented in literature and for a more detailed description of the general theory, see for example Ref. [9].

There have been several implementations of the vortex method for vertical axis turbines. For wind turbines, Strickland implemented the method to study the curved bladed turbine 1979 [10]. More recent implementations of vortex methods include the three-dimensional panel implementation by Dixon et al. [11] and the two-dimensional implementation by Deglaire et al. [12], which is based on conformal mappings instead of a panel method. One can also note the two-dimensional double wake model by Zanon et al. [13], where the leading edge separation point is determined through a boundary layer model and one additional vortex is released at each time step to create this leading edge separation. This makes the model suitable for dynamic stall simulations. The two-dimensional vortex method has also been combined with a finite element method by Ponta and Jacovkis [14], where the finite element method is used to calculate the flow close to the blade and the vortex method is used to calculate the wake.

The vortex method has been used for tidal turbines by e.g. Wang et al. [15], where the model uses a panel method for the potential flow and compensates for friction drag through a boundary layer model. Other notable work includes the work of Li and Çalişal [16], which is based on empirical values for the lift and drag coefficients, and this model also includes additional corrections for viscous effects, such as the decay of vortices. All the above mentioned models have been free vortex models, where the vortices move with the flow. One other approach is taken by McCombes et al. [17]. In this three-dimensional model, the vorticity instead is modeled on an Eulerian mesh to obtain improved stability in the solution (although a traditional free vortex model is used in the close proximity of the blades). This model is implemented through a finite volume version of the vorticity–velocity equation.

In this paper, a relatively simplified version of the vortex model will be used to ensure high computational efficiency. One simplification is that the viscous effects in the wake regions can be neglected for high Reynolds numbers. This approximation seems reasonable in most regions of the flow, apart from the boundary layer region with its large velocity gradients. A common approach is to assume that the inviscid approximation holds in the entire domain and use the Kutta condition [18] to calculate the circulation around the blade. A problem with this approach is that separation cannot be modeled, which means that stall phenomena are neglected. Another problem is that the drag force will be zero, which makes it difficult to estimate turbine efficiency. To account for this problem, profile section data is used in the following way: First, the velocities at the boundary points are calculated from the vortices (a Gaussian smoothing kernel is used in all velocity evaluations [9]). With the boundary velocities known, a linear vortex panel method is used to calculate the circulation around the blades by applying the Kutta condition (assuming zero panel vorticity at the trailing edge). To translate this into an angle of attack, a lookup table is used. The lookup table is created by calculating the circulation for known angles of attack for a stationary blade in homogeneous flow. With the angle of attack computed, the lift and drag coefficients are obtained from profile section data. With this method, a different lift coefficient is obtained than from the Kutta condition. One additional equation is necessary to calculate the circulation from these new coefficients, and following the work of Strickland [10], the Kutta Joukowski lift formula is used. The change in circulation can be calculated as the difference in circulation from the previous time step, and a vortex with a strength corresponding to this change in circulation is released into the flow each time step. The whole procedure is iterated to take into account that the released circulation changes the boundary velocity. It is assumed that the blades are small in comparison to the whole turbine, and each blade is solved individually, where the other blades are approximated as point circulations.

One limit of this model is that profile section data has to be available. For the current blade, data is obtained from Ref. [19]. For low tip speed ratios, the blades will enter the dynamic stall region. To compensate for these effects, a model originally developed by Gormont [20] and later modified by Massé [21] is used. The current implementation uses the parameters suggested by Berg [22]. The blades are assumed to have a high aspect ratio and no corrections due to 3D effects are performed.

The Kutta condition is not satisfied with the reduced blade circulation, which corresponds to infinite flow velocity at the trailing edge. To account for this, the blades are approximated as single point circulations during the vortex convection step to avoid problems at the trailing edge and to improve computational speed. Within this approximation, a few vortices may penetrate into the blade in the computational domain. Since this is unphysical, the contributions from these vortices are neglected during the calculation of the blade circulation while the vortices are inside the blade.

2.1. Modeling channels

The velocity contribution (complex conjugated) from a vortex at position z_j and with strength Γ_j in free flow is obtained at position z from

$$\overline{u(z)} = -\frac{i\Gamma_j}{2\pi}\frac{1}{z-z_j}, \tag{1}$$

where the z and z_j are complex numbers. It is shown in Ref. [23] that if the vortex is located in a channel with walls at $\mathrm{Im}(z) = 0$ and $\mathrm{Im}(z) = W$, equation (1) can be written as

$$\overline{u(z)} = -\frac{i\Gamma_j}{2\pi}\left(\coth(\sigma_c(z-z_j)) - \coth\left(\sigma_c\left(z-\overline{z_j}\right)\right)\right), \tag{2}$$

where

$$\sigma_c = \frac{\pi}{2W}. \tag{3}$$

This method ensures that there is no flow through any of the two channel walls. Equation (2) is suitable as all simulations are performed with the constant channel width W. For a more general case, a traditional panel method can be used to create the channel walls.

One method to calculate the circulation around the blade is to use a linear panel method and apply the Kutta condition [24]. A panel in this method is linear between the points z_1 and z_2 and has a strength that varies linearly between Γ_1 and Γ_2. For wide channels when the blades are far away from the channel walls, one approximation is to use the free flow expression when solving the Kutta condition (as described in Ref. [24]). This approximation holds when the size of the blade is small compared to the distance to the channel wall. If one blade is close to a wall, equation (2) has to be used to derive the exact expression for full accuracy. For consistency, the exact expression will be used throughout this work.

The expression for the flow velocity of a linear panel with linear strength can be obtained by integrating equation (2) over the panel. If equation (2) is split into two parts $\overline{u(z)} = \overline{u_1(z)} + \overline{u_2(z)}$ where