数理解析研究所講究録 1288 巻 2002 年 44-51

Numerical simulation of the flow around a cross-flow wind turbine

お茶の水女子大学大学院人間文化研究科　河村哲也，佐藤祐子
Tetuya KAWAMURA, Yuko SATO
Graduate School of Humanities and Sciences, Ochanomizu University, Tokyo

Two-dimensional flows around the cross-flow type wind turbine are investigated by numerical simulation. The turbine studied in this paper has cylindrical shape with 12 small blades along its periphery. Incompressible Navier-Stokes equation is used for this simulation. A rotating coordinate system, which rotates at the same speed of the turbine, is used in order to simplify the boundary conditions on the blades of the turbine. Additionally, a boundary fitted coordinate system is employed in order to express the shape of the blades precisely. The fractional step method is used to solve the basic equations. A third order upwind scheme is chosen for the approximation of the non-linear terms since it can compute the flow field stably even at high Reynolds number without any turbulent models.

1. INTRODUCTION

Recently, wind energy has been received much attention as a natural energy source. The wind turbine is one of the best candidates for this purpose. While the number of wind power plants for electricity have increased recently, we focus on a wind turbine that is applicable to pumping water for irrigation since we are interesting on the environmental problems in arid lands. The wind turbine of cross-flow type is suitable for this purpose due to its ability of getting high torque. Another advantage is its lesser dependence on the wind direction. Among many kinds of cross flow turbine, we look at the cylindrical turbine with many blades since we can find few investigations about this turbine and its performance is not well known. The purpose of the present study is to investigate the flow field around the cylindrical turbine numerically and to estimate the torque and power coefficients. These information give us fundamental data for the design of the turbine. Also, we intend to show the effectiveness of the numerical simulation on the study of the wind turbine.

2. NOTATIONS IN THIS STUDY

Following symbols are used in this paper. Other basic fluid dynamical quantities have their usual

R : Radius of the turbine
D : Diameter of the blade
θ : Rotation angle
ω : Angular velocity
U_∞ : Uniform wind speed far from the rotor
A : Cross sectional area of the rotor
q : Dynamic pressure = $\rho U_\infty^2/2$
T : Torque
λ : Tip speed ratio = $R\omega/U_\infty$
C_t : Torque coefficient = T/qRA
C_p : Power coefficient =λC_t
x, y : Position in the stational Cartesian coordinate
X, Y : Position in the rotating Cartesian coordinate

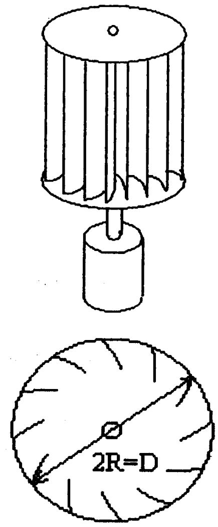


Fig. 1 Schematic figures of the cylindrical cross-flow turbine and notations

3. NUMERICAL METHOD

Since the cross-flow turbine rotates rather slowly, the flow is governed by the incompressible Navier-Stokes equations. Along the axis of the rotation (with the same angular velocity ω as the rotor), these equations become as follows:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0, \tag{1}$$

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} - \omega^2 X + 2\omega V = -\frac{\partial p}{\partial X} + \frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right), \tag{2}$$

$$\frac{\partial V}{\partial t} + U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} - \omega^2 Y - 2\omega U = -\frac{\partial p}{\partial Y} + \frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right). \tag{3}$$

where (x, y) and (u, v) are the position and velocity components in stational coordinate, (X, Y) and (U, V) are those in the rotational coordinate (see Fig.2).

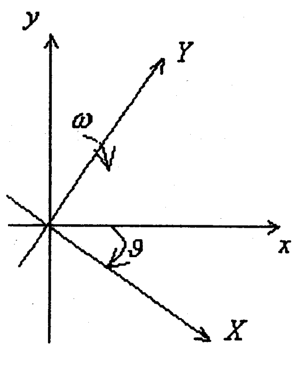


Fig. 2 Schematic figure of the relation between stational and rotational coordinate

The following relations hold between the coordinate systems:

$$x = X\cos\theta + Y\sin\theta, \quad X = x\cos\theta - y\sin\theta,$$

$$y = -X\sin\theta + Y\cos\theta, \quad Y = x\sin\theta + y\cos\theta, \tag{4}$$

$$u = U\cos\theta + V\sin\theta + \omega y, \quad U = u\cos\theta - v\sin\theta - \omega y,$$

$$v = -U\sin\theta + V\cos\theta - \omega x, \quad V = u\sin\theta + v\cos\theta + \omega x. \tag{5}$$

These equations are solved by the fractional step method[4)]. This method is summarized briefly for the present equations as follows. At first, temporal velocity components are computed by the following equations

$$u^* = u + \Delta t\left(-\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) + \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)\right),$$

$$v^* = v + \Delta t\left(-\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) + \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)\right). \tag{6}$$

The right-hand-side of the equations (6) is calculated by using the velocity components at 'n'th time step. These equations are obtained by omitting the pressure term from the original momentum equations. Then, the pressure is determined from the Poisson equation:

$$\frac{\nabla \cdot \boldsymbol{v}^*}{\Delta t} = \Delta p^{n+1}. \tag{7}$$

The new velocity components at '$n+1$'th time step are calculated from

$$u^{n+1} = u^* + \Delta t(-\nabla p^{n+1}),$$

$$v^{n+1} = v^* + \Delta t(-\nabla p^{n+1}). \tag{8}$$

This procedure is continued until it reaches to the prescribed time steps.

In order to impose boundary conditions precisely on the blades (no-slip in rotating coordinate system), the boundary-fitted coordinate system is employed. The transformed equations are solved by the finite difference method.

All the spatial derivatives except nonlinear terms are approximated by central difference. Nonlinear terms are approximated by third order upwind difference since it provides a stable solution without any turbulence models even at very high Reynolds number.

Then the torque acting on the segment of the blade can be calculated from

$$\begin{aligned} \Delta T &= y\Delta f_x + x\Delta f_y \\ &= (y\Delta y + x\Delta x)(p_2 - p_1). \end{aligned} \tag{9}$$

The total torque acting on the blade is obtained by summing up the contribution from each segment to give

$$T = \sum_{i=1}^{n} \Delta T_i. \tag{10}$$

where the parameters are as follows:

p_1, p_2 : the pressures on either sides of the blade obtained by the flow calculation

x, y : the position of the center of the segment

$\Delta x, \Delta y$: the x, y components of the length of the segment

f_x, f_y : the x, y components of pressure force acting on the segment

(see Fig.3)

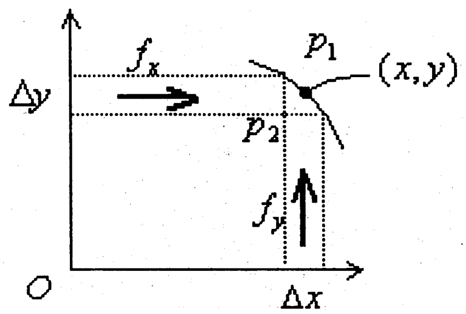


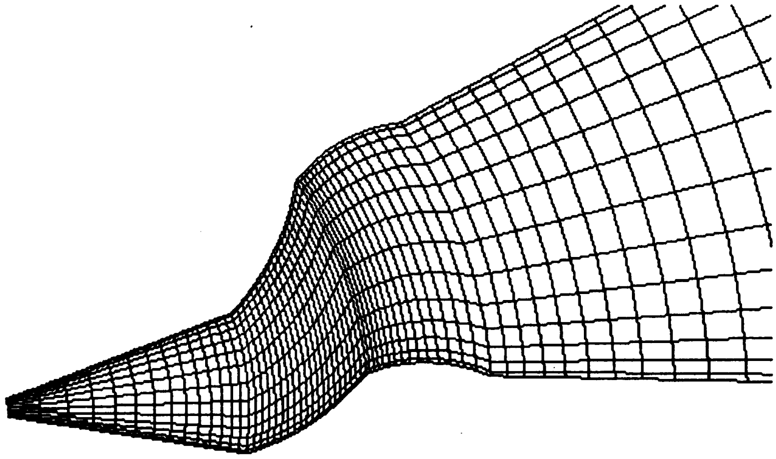
Fig. 3 Definition diagram for the calculation of torque

In this study, the cylindrical cross-flow turbine with 12 blades is tested. The shape of the cross section of the blade is a part of the circle. The thickness of the blade is assumed to be zero in this computation.

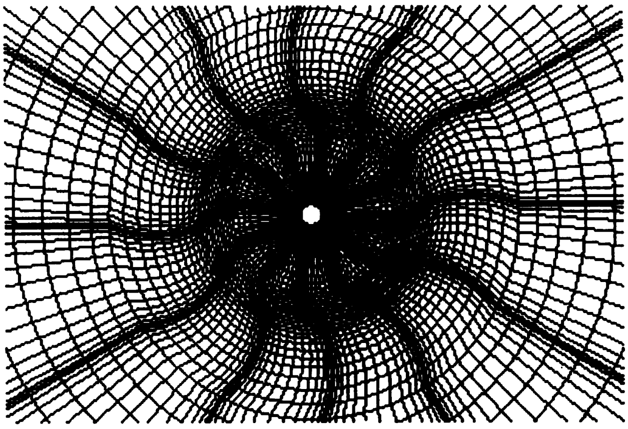
Considering the shape of the cross-flow turbine, the grid generation is enough to only one-twelfth region. At first, the grid is generated algebraically with grid concentration enforced near the blade (see Fig.4(a)). Then, the whole grid is obtained from this grid by rotating 30 (=360/12) degrees 12 times (see Fig.4(b)). The number of grid points is 92 × 184.

The boundary of each region is overlapped and the following boundary conditions are imposed on each boundary. a)On the blade (e.g. along BC in Fig.4(c)), no slip condition is applied, b)on the other boundary (e.g. along AB and CD), the velocity and pressure are replaced by average value of the neighboring grid points (e.g. P and Q) at each time step, c)on the far boundary, the velocity is assumed to be uniform and the pressure is extrapolated.

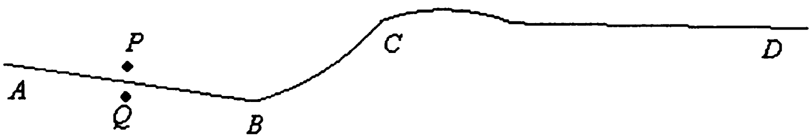
Initially, uniform velocity and pressure field are set in the whole region. Since the grid is not very fine, the Reynolds number (based on the uniform flow and diameter of the rotor) is fixed to 2000.



(a) One-twelfth region



(b) Whole grid near the blade



(c) Lower boundary of the region shown in (a)

Fig. 4 Grid generation

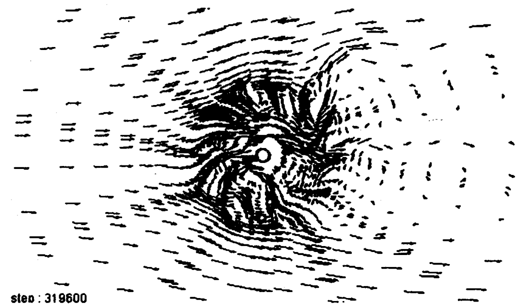
4.RESULTS

Numerical simulation of the flow around the rotating cylindrical cross-flow turbine is performed, and both the torque and power coefficients are calculated to evaluate its performance.

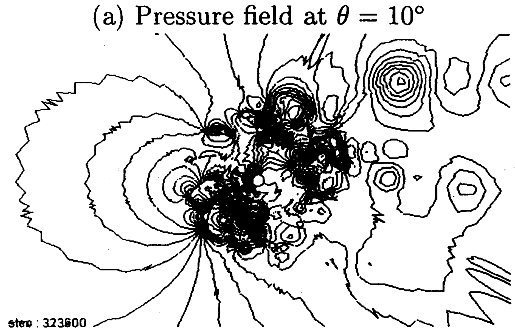
4.1 Basic Flow Field

Figure 5 shows the velocity field and the pressure contours around the cross-flow wind turbine after the flow becomes periodic. It shows the flow fields at $\theta = 10°$ and $\theta = 25°(= 10° + 15°)$ as the typical cases since the wind turbine has a period of 30 degrees. The tip speed ratio is 0.2. Roughly speaking, the flow around the turbine is similar to the one around a concentric cylinder with radius R, i.e. the pressure is high near the front side and Karman vortex can be seen behind the body. In the figure of the vel field, the flow into the turbine pushes the blades on the downstream side once again(see Fig.6). It is clear that the strong vortices are generated and shed at the edge of the blades in the upper side o turbine where the blades are pushed by the flow. These vortices flow downstream. In the lower si the turbine where the blades are moved against the flow, we can see the weak vortices.

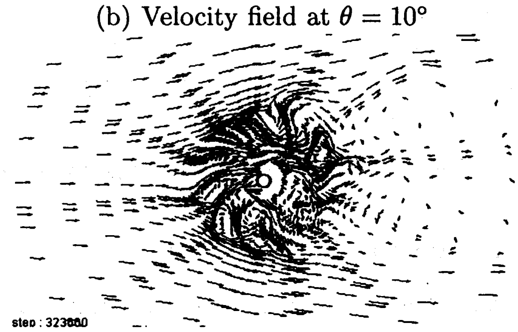
(a) Pressure field at $\theta = 10°$



(b) Velocity field at $\theta = 10°$



(c) Pressure field at $\theta = 25°$



(d) Velocity field at $\theta = 25°$

Fig. 5 Flow field near the cross-flow turbine ($\lambda = 0.2$)

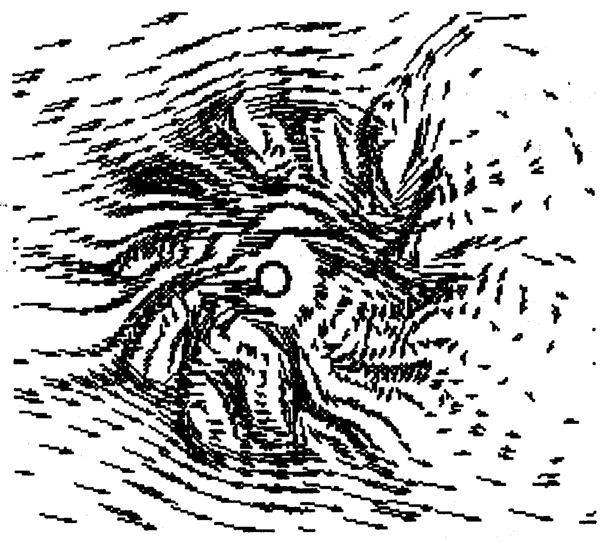


Fig. 6 Close-up of the velocity field near the blade in Fig.5(b)

4.2 The effect of the tip speed ratio

The tip speed ratio λ is an important parameter which determines the performance of the wind turbine. Figure 7 shows the effect of the tip speed ratio on both the torque and power coefficients. It is clear from this figure that the torque coefficient becomes smaller as the tip speed ratio increases while the power coefficient has its maximum value around $\lambda = 0.28$.

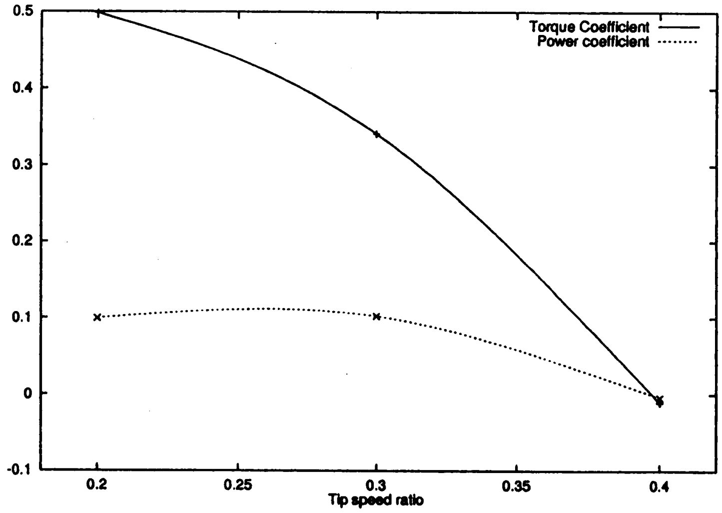


Fig. 7 The effect of the tip speed ratio on the torque and power coefficients

4.3 The effect of the guide vane

We also compute the flow field around the cross-flow turbine with the guide vanes to promote its performance. Figure 8 shows the shape and location of the guide vanes. Figure 9 is the flow field around this turbine. The guide vane on the upstream side (e.g. A in Fig.8) makes the flow slow behind the vane, which prevents the drag acting against the rotation of the turbine. The guide vane on the downstream side (e.g. B in Fig.8) causes the more fluid come inside of the turbine compared to the case without the vane. This flow helps the rotation of the turbine. We can also see the large separation vortices generated by the guide vane on the downstream side.

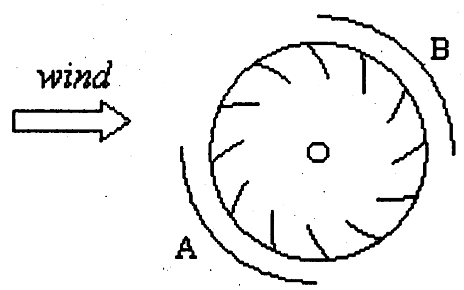
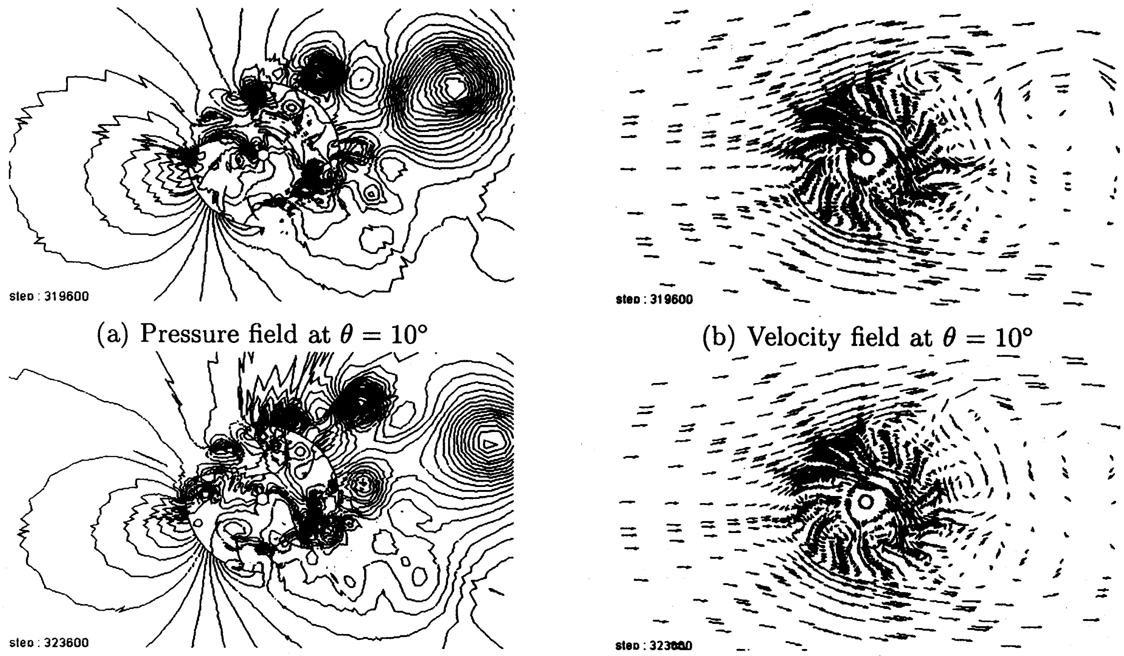


Fig. 8 The shape and location of the guide vanes



(a) Pressure field at $\theta = 10°$

(b) Velocity field at $\theta = 10°$

(c) Pressure field at $\theta = 25°$

(d) Velocity field at $\theta = 25°$

Fig. 9 Flow field near the cross-flow turbine with guide vanes ($\lambda = 0.2$)

5. SUMMARY

Owing to the demand for the energy sources not harmful to the environment, the wind power draws attention of public. The wind turbine is widely used to obtain such wind energy. In this study, we show one numerical method to calculate the complex flow fields around the cylindrical cross-flow turbine and to compute the torque and power coefficients. This kind of wind turbine is suitable for the pumping water system in arid lands. The basic equations are expressed in the rotational coordinate system and the fractional step method is used to solve these equations. The computational region is divided into small regions and connected to each other since the shape of the wind turbine is not simple in general. Not only the flow field but also the effect of the tip speed ratio on these coefficients are investigated. Moreover, the effect of the guide vane which is equipped on the turbine in order to promote its performance is examined. Although the wind turbine of cross-flow type with 12 blades are chosen, the method used in this study is applicable to calculate the flow field and compute the performance of many kinds of wind turbine.

REFERENCES

1) K.Ishimatsu, T.Shinohara and F.Takuma. : Numerical simulation for Savonius rotors (running performance and flow fields). *Trans. JSME* (B),60(569):154-160,1994.

2) B.F. Blackwell R.E. Sheldahl and L.V. Feltz. : Wind tunnel performance data for two- and three-bucket Savonius rotors. *J.Energy* , 1978.

3) S.J. Savonius. : The s-rotor and its application. *Mech.Eng.,* 53:333, 1931.

4) Yanenko,N.N. : The Method of Fractional Steps, (*Springer-Velag,* 1971)