Journal of Applied Mechanics Research
Vol. 4, No.1, pp 9-14, Summer, 2012
Received 17 Apr. 2012, Accepted 15 July., 2012

Simulation of Unsteady Flow Field around a Savonius Wind Turbine

K. Abbaspoursani, Ass. Prof., the Faculty of Mechanical Engineering, Takestan Islamic Azad University
H. Forozan, Member of Academy, the Faculty of Marine Engineering, E. Khomeini Maritime University
N. Abbaspour, B.S. Student, the Faculty of Mechanical Engineering, Takestan Islamic Azad University

E-mail: abbaspoursani@yahoo.com

Abstract: The vertical axis wind turbines such as Savonius rotor have many major advantages over the others in view of operation and construction which make the rotor more attractive to small scale power applications especially in the rural areas. This lead to some researches to conduct several experiments for investigation the effects of geometrical parameters such as blade gap size, overlap and Reynolds numbers. In this study, we focus on the flow field under unsteady conditions in a Savonius rotor and make clear the running performance by mean of a numerical simulation. This study attempts to make a visualization of flow field around a Savonius blade by solving governing equations using a CFD technique. However, the results provide a cohesive design tool that provides invaluable insight into flow field that drives the blade while providing an adequate running performance of the blade.

Keywords: Savonius blade, Unsteady flow, Performance and CFD.

شبیه‌سازی میدان جریان ناپایدار پیرامون توربین بادی ساوانیوس

کمال عباسپورثانی– استادیار گروه تبدیل انرژی، دانشکده مهندسی مکانیک، دانشگاه آزاد اسلامی واحد تاکستان

حمید فروزان– عضو هیأت علمی دانشکده مهندسی دریائی، دانشگاه علوم دریائی امام خمینی (ره) نوشهر

نیما عباسپور– دانشجوی کارشناسی، دانشکده مهندسی مکانیک، دانشگاه آزاد اسلامی واحد تاکستان

چکیده: توربین‌های بادی محور قایم، همانند روتور ساوانیوس مزایای زیادی از جهت ساخت و عملکرد نسبت به انواع دیگر توربین‌های بادی دارند که باعث می‌شود در کاربردهای تولید توان در مقیاس کوچکتر، بویژه در مناطق دورافتاده مورد توجه قرار گیرد. این برتری سبب شده که محققان زیادی کارهای تجربی مختلفی برای بررسی اثرات پارامترهای هندسی از قبیل اندازه شکاف پره، هم‌پوشانی پره‌ها و اعداد رینولدز انجام دهند. در این مقاله تأکید بر شرایط جریان ناپایدار در روتور ساوانیوس و شفاف سازی عملکرد آن توسط شبیه‌سازی عددی می‌باشد. در این مقاله نمایش هندسی میدان جریان پیرامون یک روتور ساوانیوس با حل معادلات حاکم با بکارگیری روش مکانیک سیالات محاسباتی ارائه می‌شود. نتایج نشان می‌دهد که این روش وسیاه طراحی مناسبی است که می‌تواند نگاه عمیقی به داخل جزئیات میدان جریان داشته باشد و همچنین می‌توان بکمک آن عملکرد پره را به روش مناسب بدست آورد.

واژه‌های کلیدی: تلوراید تیتانیوم سرب، پراش پرتو ایکس ، ریزساختار و پارامتر شبکه.

1. Introduction

The vertical axis wind energy conversion system (VAWECS) such as Savonius rotor never become developed similar to horizontal axis WECS, probably because of their low efficiency. However, the Savonius rotor has many major advantages over the horizontal axis WECS in view of operation and construction aspects [1-5]. These are its low cut-in speed , high starting torque, simple construction, and less expensive in fabrication and maintenance, which made the rotor be attractive to small scale power applications, especially in rural parts of developing countries.

The concept of the rotor was developed by a Finish engineer, Savonius, who made a rotor by cutting the flattener cylinders into the halves along the centurial plane [6], and then he moved the two semi-cylindrical surfaces sideways along the cutting plane so that he made the cross section resembled the letter “S”. According to the literature the rotor has been made by Savonius had a maximum efficiency of 31% and for its prototype 37%. However, some researchers who have conducted experiments with Savonius rotor have disagreed with the claimed efficiency [3, 4]. Therefore such different results lead the other researchers to investigate the performance of Savonius rotor subject the some geometry and flow parameters, such as, blade gap size, overlap, and Reynolds numbers. These investigations included wind tunnel tests, field experiments, and numerical simulations.

Blade configuration such as aspect ratio, blade overlap and gap, and the effects of adding extensions, and plates, and shielding were studied by [5-10] in wind tunnels. Detailed field experiments have been conducted by [2-5,11]. Numerical investigations from evaluating of performance characteristics and from flow visualization have done by [12, 14].

In this study, we focus on the flow field under a Savonius rotor and make clean the running performance by mean of the numerical simulation and the results are compared with experimental data published in literature. The variation of the torque and power coefficients for a prescribed tip-speed ratio have been calculated and compared with experimental results

2. Geometry Configuration

Here, a Savonius rotor with thin blades in the shape of Benesh profile is considered. This profile selected because experiments show it produce a high power coefficient in comparison with other profiles [15]. Figure (1a) shows the cross section of this rotor. As shown in fig (1b) the basic geometry is defined by the blade cord length, D, the blade overlap, S, radius of rotation, R, and the angular position of the blade, θ..

3. Governing Equations

The nature of the flow around a savonius rotor can be considered as incompressible, since the rotational frequency is low enough [13]. The basic incompressible Navier-stoke in non-rotating coordinate systems in non-dimensional form are:

$$\begin{cases} \boldsymbol{\nabla}\vec{\boldsymbol{V}} = \mathbf{0} \\ \dfrac{\partial\vec{\boldsymbol{V}}}{\partial\boldsymbol{t}} + \boldsymbol{V}.\vec{\boldsymbol{\nabla}}(\vec{\boldsymbol{V}}) = -\vec{\boldsymbol{\nabla}}\boldsymbol{p} + \dfrac{\mathbf{1}}{\boldsymbol{Re}}\boldsymbol{\nabla}^{\mathbf{2}}\vec{\boldsymbol{V}} \end{cases} \qquad (1$$

)

Where $\vec{\boldsymbol{V}}$, is the velocity vector and Re is the Reynolds number. By denoting fixed coordinate system (X,Y) and the rotating coordinate system (x,y) which rotates around vertical axis with a constant angular velocity of ω. Considering configuration of Fig (1b), then thus systems can be transformed to each other by followings:

$$\begin{cases} \boldsymbol{x = X\cos\theta + Y\sin\theta} \\ \boldsymbol{y = X\sin\theta + Y\cos\theta} \end{cases} \quad (2)$$

Where,θ is the angular position of rotating coordinate system fixed the blade, by taking the first time derivative of Eq. (2) has been obtained:

$$\begin{cases} \boldsymbol{u = U\cos\theta - V\sin\theta - \omega y} \\ \boldsymbol{v = U\sin\theta + V\cos\theta + \omega x} \end{cases} \quad (3)$$

Where (u, v) and (U, V) are the velocity components in rotating coordinate and non – rotating coordinate systems, respectively. Using the transforms of Eqs (2) and (3), the governing equations of (1) are expressed in the rotating coordinate systems as:

$$\begin{cases} \boldsymbol{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0} \\ \boldsymbol{\frac{Du}{Dt} - \omega^2 + 2\omega v = -\frac{\partial p}{\partial x} + \frac{1}{Re}\nabla u} \\ \boldsymbol{\frac{Dv}{Dt} - \omega^2 - 2\omega u = -\frac{\partial p}{\partial y} + \frac{1}{Re}\Delta v} \end{cases} \quad (4)$$

The boundary conditions associated with the above governing equations are assumed to be free stream condition and initially is zero).

4. Computational Method

To provide a knowledge concerning the nature of the flow field around the rotating blade, the above governing equations associated with boundary and initial conditions have been solved by CFD technique by using ver. 6.0 of the fluent, software program. The simulation was performed in a 2-D space domain since the blade is inherently a 2-D device. To do this the computation domain including the rotating rotor and air around it is divided into same small grids. The manner in which the cells are arranged is shown in Fig.2. The simulation model used the Reynolds-stress turbulence model because the K-ε model cannot accurately simulate the follow separation region that occurs at the tips of the blades. Cochran et.al have shown that the simulation model using k-ε method predict a strong vortex that the overall efficiency of blade should rotate backward [14]. Therefore the k-ε model is unreliable form this application, and hence we use the Reynolds-stress model. As show in Fig.2, the structure of the cells used within the simulation model in a non-uniform mesh not only the number of cells is important, but also the orientation and placement of the cells properly must be defined. Experiments show that as the blade rotates, there is a large increase in flow velocity over a short time[], therefore in the region of the blade walls and near the tip blade there is a need for small cells, as shown in Fig(2).

Simulations were performed using a grid with a mesh size over 15000 cells and repeated with a finer mesh featuring over 9718 cells, and the obtained results were compared which show the results matched closely. This indicates that the obtained results with a mesh over 9718 triangular cells are grid independent. The simulation runs begin by assuming that the whole computation domain has initially a velocity same as the inlet. Generally, the flow field around the blades includes an accelerated flow region around the blade edges and a low velocity wake behind them[14], so to capture this phenomenon accurately, the time step size used with this simulations is chosen as 0.1 are performed with marching time technique to establish a general flow field around the rotor.

5. Results

Typical results associated with the performance of the simulation are described here. The overlap of 2 bunked blades $\left(\frac{\mathbf{S}}{\mathbf{D}}\ \textbf{in Fig. (1)}\right)$is chosen as 0.33, corresponding to the maximum blade efficiency, and assumed a tip speed ratio $\left(\frac{\mathbf{R\omega}}{\mathbf{U_\infty}}\right)$ as 1.1. Figure (3) shows the instantaneous velocity vectors and the mean velocity vectors and the mean velocity contours for both the inner and outer regions of the blade over 180° rotation. As can be seen in beginning of the flow (zero angular position), there is a very large region of flow separation and re-circulation just aft of maximum chamber location, which results in large momentum loss during its operation, also there is a small separated region at the upper surface of the blade near the leading edge, but the flow quickly reattaches after that.

Figure(4) is similar to Fig(3), but it is from 45° angular position which indicates an increase if flow separation in the upper side of the blade. However the amount of separation in the lower side is reduced and ad shown in Fig(5) at 90° angular position it is significantly reduced which should results an increase in rotor overall efficiency.

Figures (6) and (7) show the instantaneous velocity vectors and the mean velocity contours at 135° and 180° angular positions, respectively. By a careful examination of these figures it reveals that as blades passes from 90°angular position, there is a large flow separation and re-circulation regions which cause a loss of momentum which should result in lower performance. An improvement id expected when the rotor passes from 180° angular position.

6. Conclusions

In this study, the flow field around a Savonius rotor with Benesh blade is observed by solving governing equations using CFD software. However the results provides a cohesive design tool that provide invaluable insight into flow field that drives the rotor while providing an adequate turning performance of the rotor the obtained result shoe that there is a large flow separation and re-circulation regions in the upper surface of the blade at downstream which cause in lower performance of the rotor .

7. References:

[1].Fernando, M.S.U.K and Modi, V.J: 1989 “A Numerical Analysis of the Unsteady Flow Past a Savonius Wind Turbine”, *J. of Wind Engineering and Industrial Aerodynamics*, Vol.32, pp303-327.

[2].Mojola, o.o.:1985, “On The Aerodynamics Design of the Savonius Windmill Rotor”, *J of Wind Engineering and industrial aerodynamics*, Vol.21, pp221-310.

[3].Fujisawa,N. and Gotoh, F.,: 1994, “Experimental Study on the Aerodynamic Performance of a Savonius Rotor”, *ASME J. of Solar Energy Engineering*, Vol.116, pp136-49.

[4].Huda, M.D., Selim, M.A., Islam, A.K.M.S., and Islam M.A.: 1992, “The Performance of an S-shaped Savonius Rotor with a Deflecting Plate”, *RERIC Int. Energy J.*Vol.14, pp25-32.

[5].Ogawa, T., Yoshida, H., and yokata,Y.: 1989, “Development of Rotational Speed Control System for a Savonius-Type Wind Turbine”, *ASME J. of Fluids Engineering*, Vol.III, pp53-8.

[6].Fujisawa, N. and Gotoh, F,:1992, “Pressure Measurement and Flow Visualization Study of Savonius Rotor”, *J. of Wind Engineering and Industrial Aerodynamics*, Vol.39, pp51-60.

[7].Ushiyama, I, Nagai, H., and Mino,M.: 1982, “The Optimum Design Configurations of SavoniusWind Turbines”, *Proc. Of 17Th Intersociety Energy Conversion Engineering conf.*, pp2089-195.

[8].Saylers, A.T: 1995, “Blade Configuration Optimizations and Performance Characteristics of a Simple Savonius Rotor”, *Proc. of Institution of Mechanical Engineers*, Vol.199, pp185-91.

[9].Sheldahl, R.E., Blackwell, B.F., and Feltz, L.V.: 1978, “Wind Tunnel Performance Data for Two

and Three Bucket Savonius Rotor", *J. of Energy*, Vol.2, pp160-4.

[10]. Alexander, A.J. and Holowina, B.P.: 1978, "Wind Tunnel Test on a Savonius Rotor", *J. of Industrial Aerodynamics*, Vol.3. , pp343-51.

[11]. Modi, V.J. and Fernando, M.S.O.K.: 1989, "On the Performance of the SavoniusWind Turbine", *ASME J. Solar Energy Engineering*, Vol.III, pp71-81.

[12]. Saha, U.K. and Rajkuman, M.I: 2006, "On the Performance Analysis of Savonius Rotor with Twisted Blades", *J. of Renewable Energy*, Vol.31, pp1776-88.

[13]. Kawamura, T., Hayashi, T., and Miyashita, K.: 2001, "Application of the Domain Decomposition Method to the Flow around the Savonius Rotor", *Proc. Of the 12th int. conf. on Domain Decomposition Methods*, pp393-400.

[14]. Cochran, B.C. , Banks, D. , and Taylor, S.I.: 2004, "A Three-Tiered Approach for Designing and Evaluating Performance Characteristics of Novel WECS", *AIAA 2004*, Vol.1362, pp1-11.

[15]. Khan, M.H.: 1978, "and Prototype Performance Characteristics of Savonius Rotor Windmill", *Wind Engineering,*Vol.2, pp75-85.

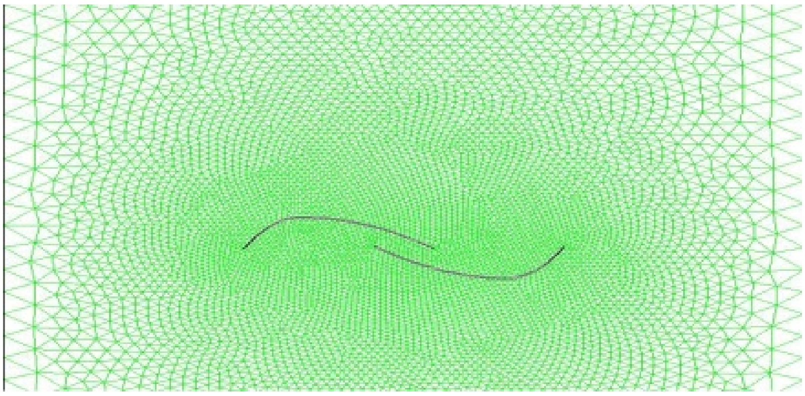


Fig. (1): Geometry of a Savonius blade

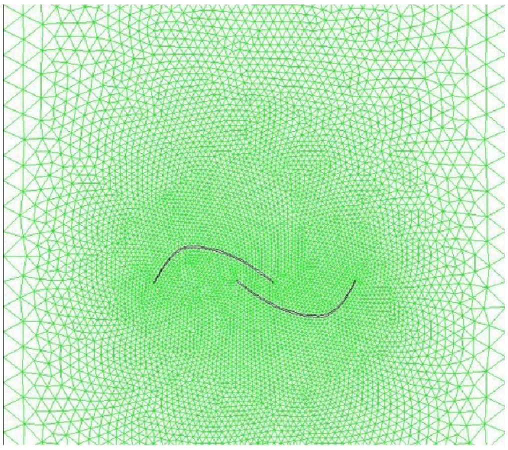
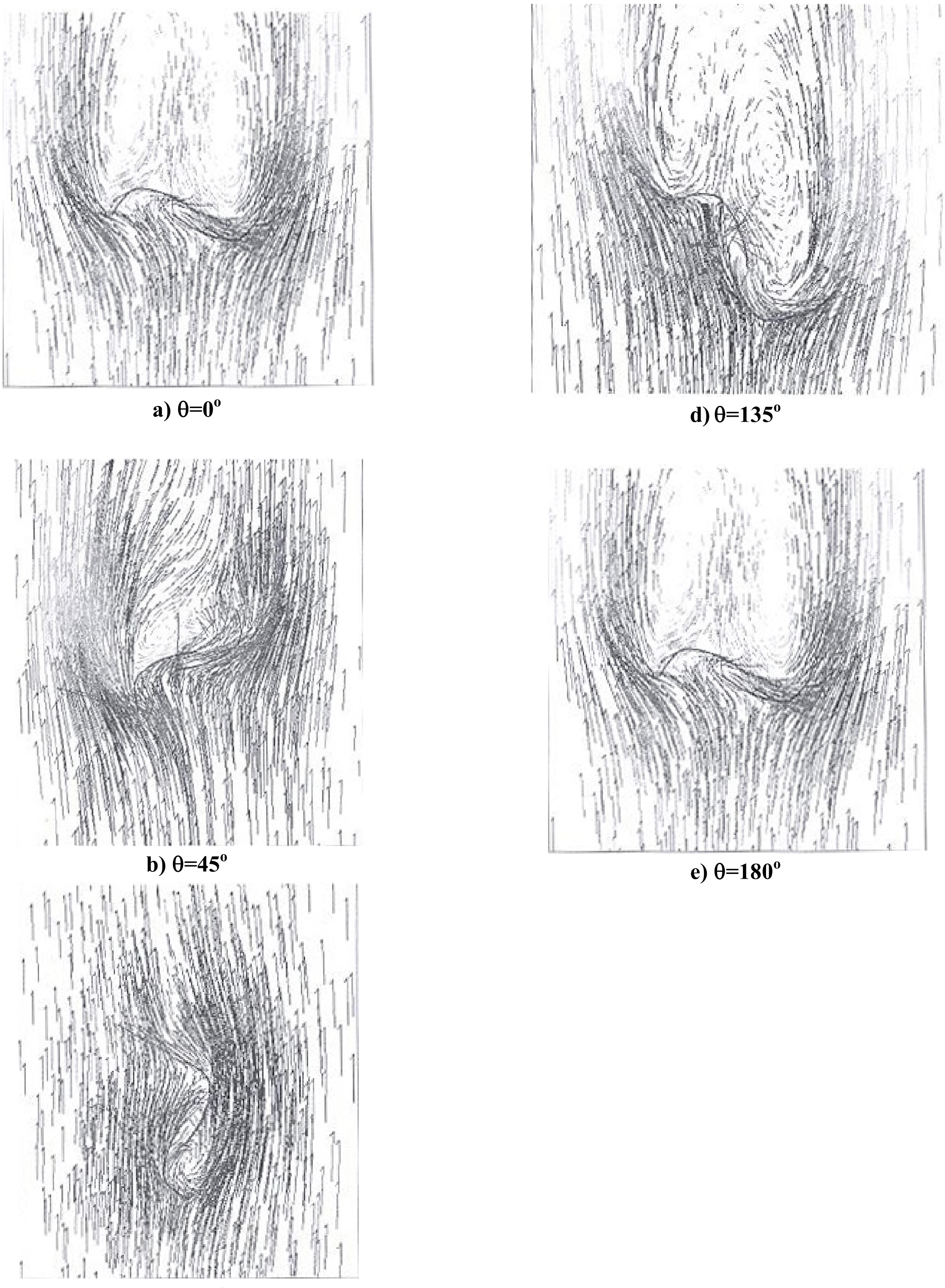


Fig. (2): Grid of computational domain



a) θ=0°

d) θ=135°

b) θ=45°

e) θ=180°

c) θ=90°

Fig. (3): Field flow around the blade and velocity vectors at various angular position