Journal of Energy Technologies and Policy
ISSN 2224-3232 (Paper) ISSN 2225-0573 (Online)
Vol.3, No.1, 2013

Simulation of Flow Pattern through a Three-Bucket Savonius Wind Turbine by Using a Sliding Mesh Technique

Dr. Dhirgham AL-Khafaji

Department of Mechanical engineering/College of Engineering, University of Babylon/ Babylon-Iraq

Abstract

Vertical axis wind turbine (VAWT) type Savonius rotor is self-starting, inexpensive, less technicality & high productivity wind machine, which can accept wind from any direction without orientation, and provides high starting torque. The investigations of aerodynamic parameters and the flow pattern of the turbulent flow through the rotor have high aspect on Savonius wind turbine performance. The flow pattern through a three buckets Savonius rotor model of 10cm diameter inside smoke wind tunnel with high-speed camera was investigated experimentally. The commercial code FLUENT 6.3.26 used to simulate the turbulent flow ($M<0.3, Re>2000$) by RNG $K-\varepsilon$ turbulent model. Two-dimensional model carried out the simulation of the flow pattern, velocities and pressures of airflow within a Savonius wind rotor placed in Smoke wind tunnel. The domain of the airflow dived for two zone, first zone up and down stream flow meshed by fixed structured grid generation. The second zone is around Savonius rotor; the flow pattern created by a pitched blade rotor was calculated by using a sliding mesh technique with unstructured grid generation. Three time steps between two blades of rotor is taken, which give three angular orientations of blades.

The CFD results show good agree with experimental results of flow pattern. It is concluded that the sliding mesh method is suitable for the prediction of flow patterns around wind turbine. Then after ensured from the reliability of CFD simulation, it can be used for studying the velocity contour and the pressure distribution around the turbine.

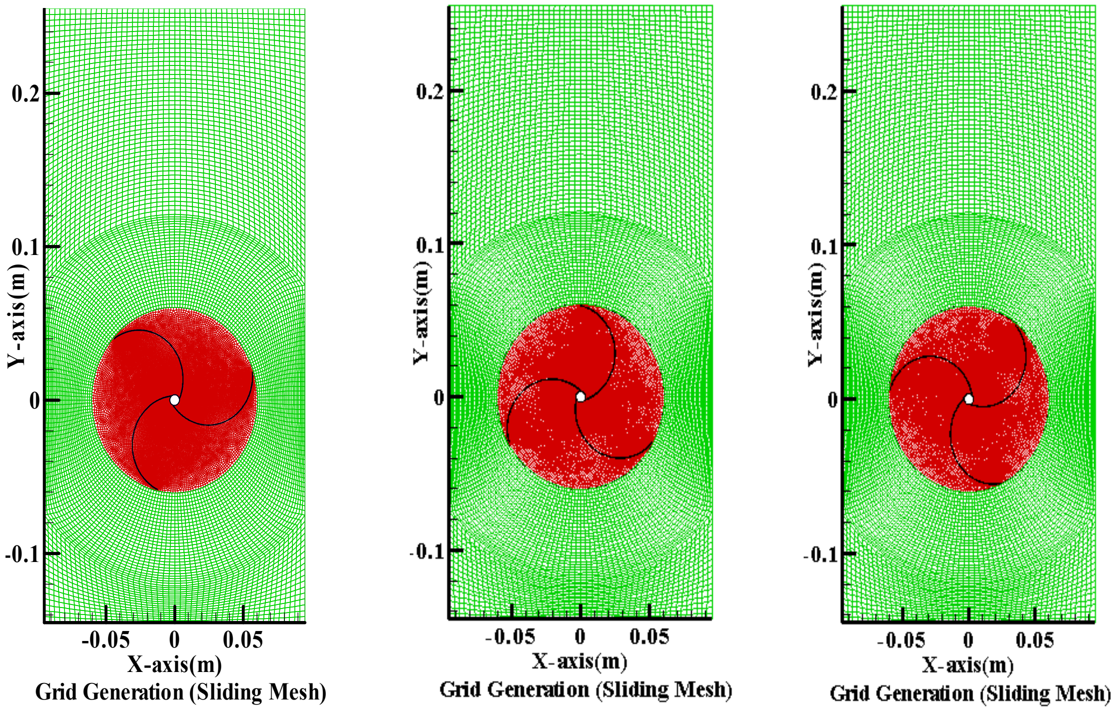
1.Introduction

Computational fluid dynamics models are now regularly used to calculate the flow patterns in the Savonius wind rotor turbine. To model the turbine, it is common to prescribe experimentally obtained velocity data in the outflow of the turbine, see e.g. Bakker and Van den Akker [1]. This has the disadvantage that it is often necessary to extrapolate the data to situations for which no experiments were or can be performed. Only recently have methods become available to explicitly calculate the flow pattern around the turbine blades without prescribing any experimental data. The sliding mesh method is a novel way of dealing with the turbine blades-wind interaction. The main advantage of the sliding mesh method is that no experimentally obtained boundary conditions are needed, as the flow around the turbine blades is being calculated in detail [2]. This allows modeling of Savonius wind rotor turbine systems for which experimental data is difficult or impossible to obtain. The purpose of this paper is to report on initial studies to the suitability of this novel method for the prediction of the flow pattern, velocity contour, and the pressure distribution through the wind turbines. We will first discuss the sliding mesh method, then present computational results and a comparison with experimental data, and finally the simulated velocity contour and pressure distribution will be discussed over a three time steps of three blade Savonius wind turbine.

2. Sliding Mesh Method

Dynamic sliding mesh simulations are required in several engineering applications such as wind turbine analysis and require an efficient implementation of non-conformal mesh joining. A variety of capabilities to handle such mesh configurations exist and include finite element-based multipoint constraint methods [3], interface capture and tracking methods [4] and cell centered finite volume methods (CCFVM) in which a flux matching protocol is employed [5], where care is required to reconstruct a conservative area representation between the two blocks [6].

With the sliding mesh method the wind tunnel is divided in two regions that are treated separately:, first zone up and down stream flow meshed by fixed structured grid generation, The second zone is around Savonius rotor; the flow pattern created by a pitched blade rotor was calculated by using a sliding mesh technique with unstructured grid generation. See Figure 1



a b c

Figure 1. Grid Used in The Sliding Mesh Method With Different Time Steps: (a) at Time (t)=0.02062 s, (b) at Time (t) =0.08247 s, (c) at Time (t) =0.1649 s.

3.Turbulent Sliding Mesh

In the wind tunnel region the standard conservation equations for mass and momentum are solved. In the rotating turbine region a modified set of balance equations is solved:

$$\partial/(\partial x_j)(u_j - v_j) = 0 \quad \text{……………..…….. (1)}$$

$$\partial/\partial t\, p u_i + \partial/(\partial x_j)\, p(u_j - v_j)\, u_i = -\partial p/(\partial x_i) + (\partial \tau_{ij})/(\partial x_j) \quad \text{………………...……. (2)}$$

Where: u is the wind velocity in a stationary reference frame
v is the velocity component arising from mesh motion
p is the pressure and
τ_{ij} is the stress tensor

The first equation is the modified continuity equation and the second equation is the modified momentum balance. At the sliding interface a conservative interpolation is used for both mass and momentum, using a set of fictitious control volumes. No-slip boundary conditions are used at the turbine blades, the shaft, and the tunnel walls. No experimental data is prescribed in the outflow of the turbine. All fluid motion strictly arises from the rotation of the turbine blades. The grid was generated with a proprietary program named GSMMBIT 2.2.30. The total number of mesh was approximately 50000. All simulations were performed using Fluent™ from Fluent, Inc. More details of the numerical methods can be found in Murthy *et al.* [7] and in reference [8].

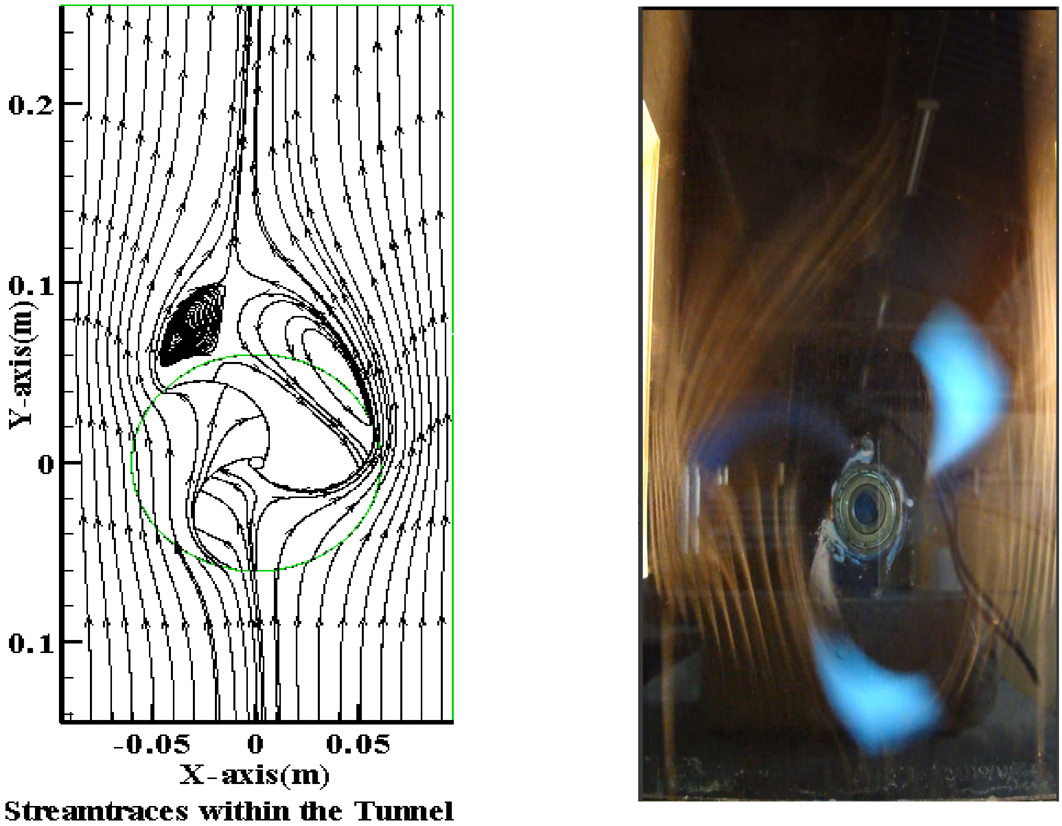
4. Simulation Design

Time dependent simulations were performed for the flow created by a pitched blade turbine in a wind tunnel with a diameter of T=10cm. The turbine diameter was D = T/3. The blade width was W = 0.2D. The turbine rotational speed was $N = 4.68 s^{-1}$ and the viscosity was fixed to obtain turbine Reynolds numbers ($Re = \rho.N.D.2/\mu$) more than Re = 2000. In this range the flow was turbulent. In that case the k-ε RNG turbulence model was used [3]. In the simulations a three time steps at t=0, t=0.08247 s, and at t=0.1649 s were used, resulting in 46.8 revolutions. Local and average velocities were tracked as a function of time to determine when periodic steady state was reached. The local velocities close to the turbine converged fastest, while the average tangential velocity in fluid bulk converged slowest.

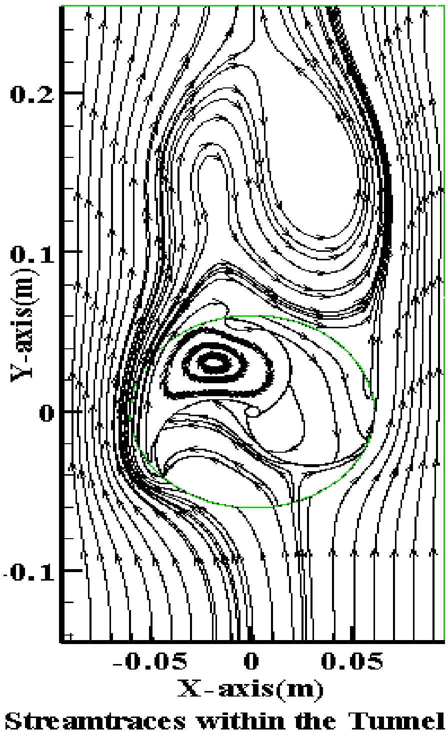
5. Experimental validation of CFD Simulation

To verify the CFD simulation results it should be compared the results from simulation with the experimental results. Figure 2 shows the results of velocity field and the stream lines that found experimentally by smoke tunnel and theoretically by simulation for three different time steps. The flow pattern is shown by means of velocity vectors. The vectors point in the direction of the air velocity at the point where they originate. The experimentally measured velocities are shown on the right while the sliding mesh results are shown on the left. At this Reynolds number the turbine creates a mainly radial flow pattern. Two-circulation loops form, above and below the turbine. The flow is very weak away from the turbine. The model results can be seen to compare quite well with the experimental visualization.

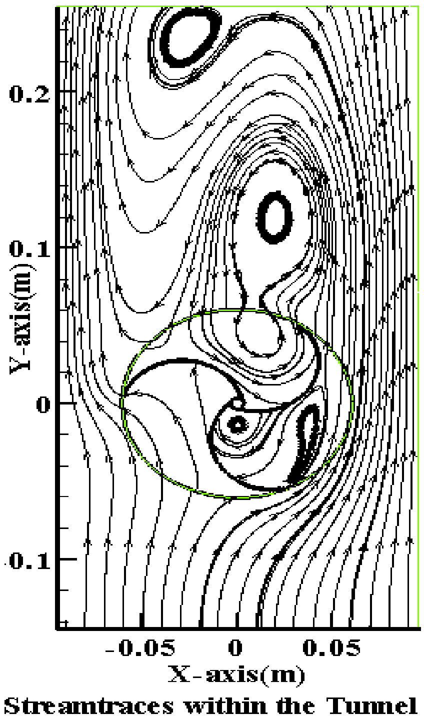
ISSN 2224-3232 (Paper) ISSN 2225-0573 (Online)
Vol.3, No.1, 2013



(a)


0.2
0.1
0
-0.1
Y-axis(m)
-0.05
0
0.05
X-axis(m)
Streamtraces within the Tunnel

(b)



(c)

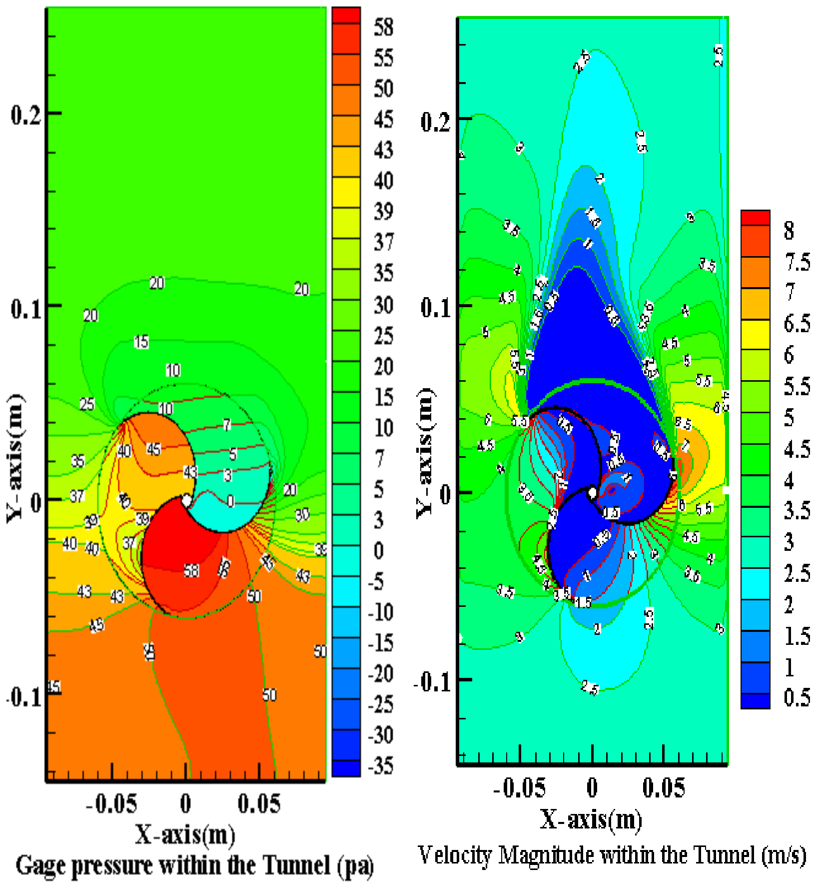
Figure 2. Comparison Between Experimental Data (right) and Sliding Mesh Results (left) With Different Time Steps: (a) at Time (t)=0.02062 , (b) at Time (t) =0.08247 s, (c) at Time (t) t=0.1649 s.

5.Simulation Results of Velocity Contour and Pressure Distribution

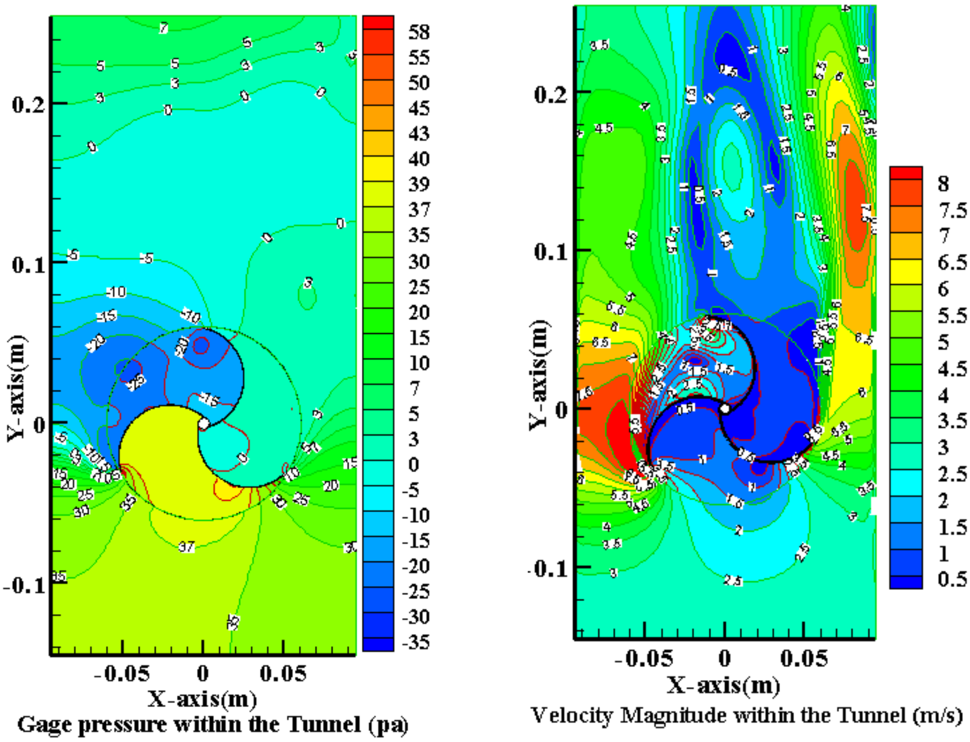
After it is ensured from the reliability of CFD simulation results, that the experimental results were in good correlation with theoretical model that was simulated with FLUENT software, so that it can be used for simulate the velocity contour and pressure distribution around the Savonius turbine without needing to the experimental data.

This study appears the two dimensional turbulent fluid flow RNG k-ε models that are taken to simulate the velocities and pressures for three time steps within Savonius Wind Turbine by using FLUENT software. The simulations compromises the analysis of the flow in the upstream, downstream of scoops and between the scoops by solving the continuity and momentum equations for incompressible Air flow through moving frame at unsteady state.

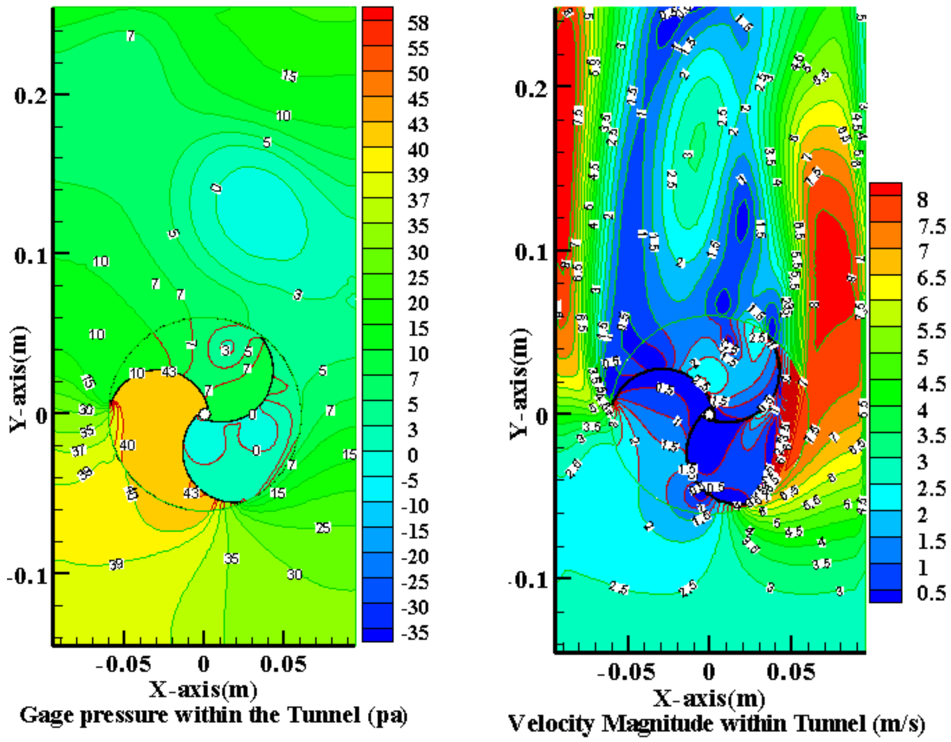
Figure 3 shows the local velocity magnitude and the pressure distribution in a plane down stream and up stream of the turbine blade. It's found that the velocity magnitude alternate as pressure distribution as according to the location from the turbine blades for the three selective time steps. Also the velocity intensity increases as the time increase while the pressure distribution decrease as the time step increase.



(a) At First Time Step (t=0.02062 s)



(b) At Second Time Step (t=0.08247 s)



(c) At third time step (t=0.1649 s)

Figure 3 Simulated velocity magnitudes (right) and simulated pressure distribution (left) with different time steps: (a) at time (t) =0.02062 , (b) at time (t) =0.08247 s, (c) at time (t) t=0.1649 s.

6.Discussion

Sliding mesh methods can be used to accurately predict the time dependent turbulent flow pattern in wind turbine of Savonius rotor type, without the need for experimental data as turbine boundary conditions. A drawback is the long calculation time, which is about an order of magnitude longer than with steady state calculations based on experimental turbine data. Furthermore, grid dependency studies will have to be performed to determine the minimum grid resolution necessary to resolve turbulent tip vortices. An important application for the sliding mesh method might be the development of new, optimized wind turbine designs for specific industrial applications and prediction of flow patterns, velocity magnitude, and pressure distribution within the turbine for which no experimental data are available.

Refrences

[1] Bakker A., Van den Akker H.E.A., (1994). Single-Phase Flow in Stirred Reactors Chemical Engineering Research and Design. *Trans. Chem.* 72, A4, 583-593.

[2] André Bakker, Richard D. LaRoche, Min-Hua Wang, and Richard V. Calabrese. (2000).

Sliding Mesh Simulation of Laminar Flow in Stirred Reactors., [Online] Available: http://www.bakker.org/cfm.

[3] Gartling, D. (2005). Multipoint constraint methods for moving body and non-contiguous mesh simulations. *Int. J. Numer. Meth. Fluids*, 47, 471-489.

[4] Tezduyar, (2001). T. E. "Finite element methods for ow problems with moving boundaries. *Comp. Meth. Engrg.* 8, 83-130.

[5] Lilek, Z, Muzaferija, S., Peric, M, S., & Seidl, V. (1988). An implicit finite-volume method using non-matching blocks on structured grid. *Num. Heat Transfer Part B*, 32, 385-401.

[6] Moen, C.D., Hensinger, D. M. & Cochran, B. (2000). Consistent areas for thermal contact between non-matching unstructured meshes. *Extended abstract for the Aerospace Sciences meeting, Reno, NV, January*.

[7] Murthy J.Y., Mathur S.R., Choudhury D. (1994). CFD Simulation of Flows in Stirred Tank Reactors Using a Sliding Mesh Technique Mixing. *Proceedings of the Eighth European Conference on Mixing, Institution of Chemical Engineers, Symposium Series* 136, 341-348, ISBN 0 85295 329 1.

[8] Fluent User's Guide, Fluent, Inc., (1995).