Methodology for Numerical Modeling the Performance of Vertical Axis Wind Turbines

Ahmed Ahmedov Krasimir Tujarov Gencho Popov

Methodology for Numerical Modeling the Performance of Vertical Axis Wind Turbines: *This paper presents a methodology for developing a numerical simulation procedure regarding vertical axis wind turbines Savonius type. Therefore the mains steps of developing the CFD analysis have been introduced – creating a geometrical model, generating a computational mesh, solver setup and carrying out the modeling.*

Key words: *ANSYS Fluent, VAWT, Numerical Modelling, y^+ Criteria, Turbulence Model.*

INTRODUCTION

The aerodynamics of the Vertical Axis Wind Turbines (VAWT) is characterized by its pronounced unsteadiness, mainly due to the constantly changing angular position of their blades during the machine operation. This is the main reason for the constant changes in the values of the relative velocity of the air flow acting on the rotor blades and the Reynolds number. The complex unsteady flow through the rotor of a VAWT at the majority of the cases is impossible to be investigated with the classical aerodynamic models like the streamtube and the vortex models [1, 2]. This problem can be avoided by using a numerical modeling approach – Computational Fluid Dynamics (CFD). The CFD modeling allows the precise modeling of the flow (vortex structures, three-dimensional effect, strut influence etc.). The conduction of a CFD modeling provides data about the flow such as, velocity fields, pressure, temperature etc. Also it allows us to visualize these results by using multicolor fields, isolines/surfaces, visualizing the flow trajectories by uncontentious lines or vector fields. The results acquired from a CFD modeling can be compared to those obtained by experimental study in an aerodynamic channel, thus leading to significant reduction in the expenses for experimental investigations. CFD modeling is considerably computational expensive approach. Even when investigating a relatively simple problem the hardware and computational time demands are high. Furthermore the accuracy of the results is hard to be evaluated in the absence or scarce of experimental data related to the studied problems.

Aim and Tasks

The aim of the present paper is the development of a methodology for numerical modeling the operation of a Savonius VAWT with two semicircular blades by the physical models included in the CFD software ANSYS Fluent 14.0.

For achieving of the aim the following tasks have been solved: an adequate two dimensional model simulating the operation of a rotating turbine have been developed; a simulation of the flow passing through the rotor is carried out; the performance characteristics of the turbine have been obtained.

The main geometrical dimensions of the investigated turbine are presented on fig. 1 and in table 1.

Computational Domain

To solve the first task (developing an adequate 2D model) the Sliding Mesh technique has been used, which leads to creating of a computational domain consisted of two separate zones. The first is a rectangular stationary outer zone with a circular opening which center matches with the center of rotation of the turbine. The second one is circular inner zone in which the geometry of the turbine is situated and rotates with the angular velocity ω of the wind rotor.

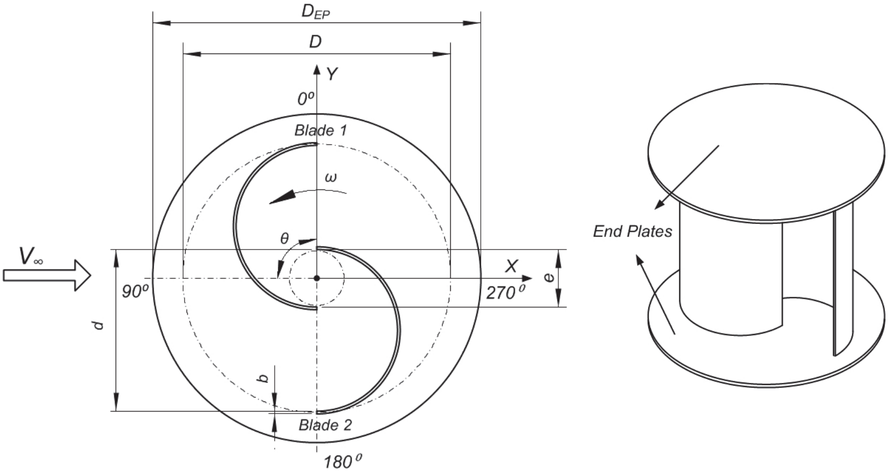


Fig. 1 Scheme of the Savonius rotor

Table 1

Main geometrical parameters of the investigated turbine		
Diameter	**D**, m	0.1
Rotor High	**H,** m	0,1
Eccentricity	**e,** m	0.35
Number of Blades	**N**	2
Blade Thickness	**b**, m	0,001
End Plate Diameter	**D_{EP}**, m	0.03
Blade Diameter	**d**, m	0.06

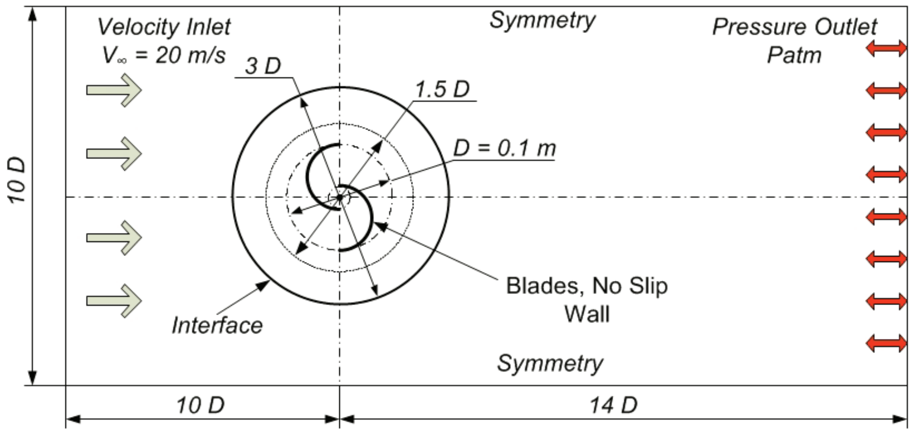


Fig. 2 Computational Domain size and Boundary Conditions

The computational domain is shown on fig. 2. The inlet and outlet sections of the domain are placed accordingly 10 diameters in front and 14 diameters behind the rotor, which according to the investigation of Ferreira et al [9, 10] allows the full development of the flow. The starting angular position is given by blade 1 as shown on fig.1. On the two horizontal walls of the computational domain a Symmetry boundary condition is applied.

On the circular wall an Interface boundary condition is applied, which provides the continuity of the flow from the outer to the inner computational zone.

Inner computational zone

The diameter of the inner computational zone is three times bigger than the rotor diameter, fig. 3. This provides enough space around him for adequate modeling of the computational mesh. Inside the inner computational zone the rotor is surrounded by a control circle with diameter 1.5D (0.150 m). In contrast with the Interface boundary condition the boundary of the control circle has no physical influence on the flow its only purpose is to provide precise control over the computational mesh in the near rotor area. This control is achieved by applying Sizing Functions which operates in direction from the blade surface towards the control circle and functions acting from the control circle towards the whole inner computational domain. The boundary condition Interior is applied over the surface of the control circle, which provides undisturbed mesh generation on both sides of the circle.

Computational Mesh Generation

For the both zones of the 2D model an unstructured computational mesh has been generated. According to an investigation carried out be Cummings [5] this type of mesh provides consistent accuracy in modeling the rotation of the turbine. Main advantages of the unstructured mesh are its simple handling and excellent application in describing complex geometries. On fig. 3 and 4 the computational mesh used in the present study is presented. For the outer stationary zone an unstructured quadrilateral mesh has been used, while for the inner zone an unstructured triangular mesh has been used. Providing a mesh with the same parameters on both sides of the Interface area leads to faster solution convergence [4]. The computational cells Growth Factor applied inside the control circle and as well as outside of it is set to be 1.2. This provides a gradual increase in the computational cells size in direction away from the rotor. The near blade mesh is controlled by the use of a Sizing Functions. A total of three Sizing Functions have been used inside the control circle: the first one is applied over the convex and concave sides of the blades and it provides cells with length $\Delta x = 1\ mm$; the second one is applied over the blades end edges and it provides cell length $\Delta x = 0.25\ mm$; the third function is applied over the control circle and it allows the generation of cells with length $\Delta x = 1\ mm$. The accuracy of the solution highly depends from the proper modeling of the laminar sublayer over the surface of the blades. In the area of the boundary layer a refined structured quadrilateral mesh has been used. The sizes of the computational cells used in the different areas of the computational domain are shown in table 2.

The $\boldsymbol{y^+}$ criteria have significant effect over the quality of the mesh and the performance of the turbulence model. This non-dimensional parameter characterizes the distance from the wall (the blade) to the first layer of cells. The near wall criteria is given by:

$$y^+ = \frac{\rho U_\tau y}{\mu} \quad (1)$$

where ρ is the air density, $\boldsymbol{y}$ is the normal distance from the wall to the first computational node from the mesh, $U_\tau = \sqrt{\tau_\omega / \rho}$ is the frictional velocity, $\tau_\omega = \mu(\partial u / \partial y)$ is the near wall tangential stress, defined with the near wall velocity gradient in normal direction, μ is the air dynamic viscosity.

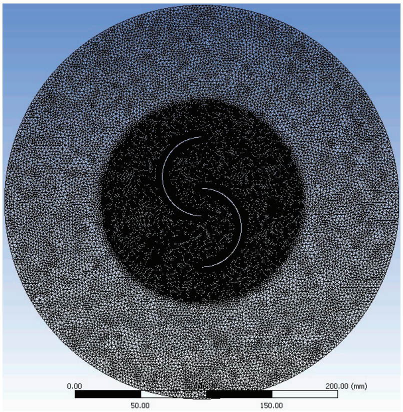


Fig. 3 Unstructured triangular mesh in the near rotor area

Table 2

Stationary Outer Domain	
Maximum Size	10 mm
Size at the Interface Area	2.5 mm
Inner Rotational Domain	
Maximum Size	2.5 mm
Size in the Near Blade Area	1 mm
Cell Length on Blade Surface	1 mm
First Cell Row High	0.01 mm
Growth Factor	1.2

The precision of the numerical modeling is determined by the value of the $\boldsymbol{y^+}$ criteria:

- $30 < y^+ < 300$ this range is recommended for simulations with activated wall function, in these cases the mesh allows flow modeling only to the turbulent region $y^+ > 30$.
- $1 < y^+ < 5$ these values are typical for meshes fine enough to allow modeling of the boundary laminar sublayer.

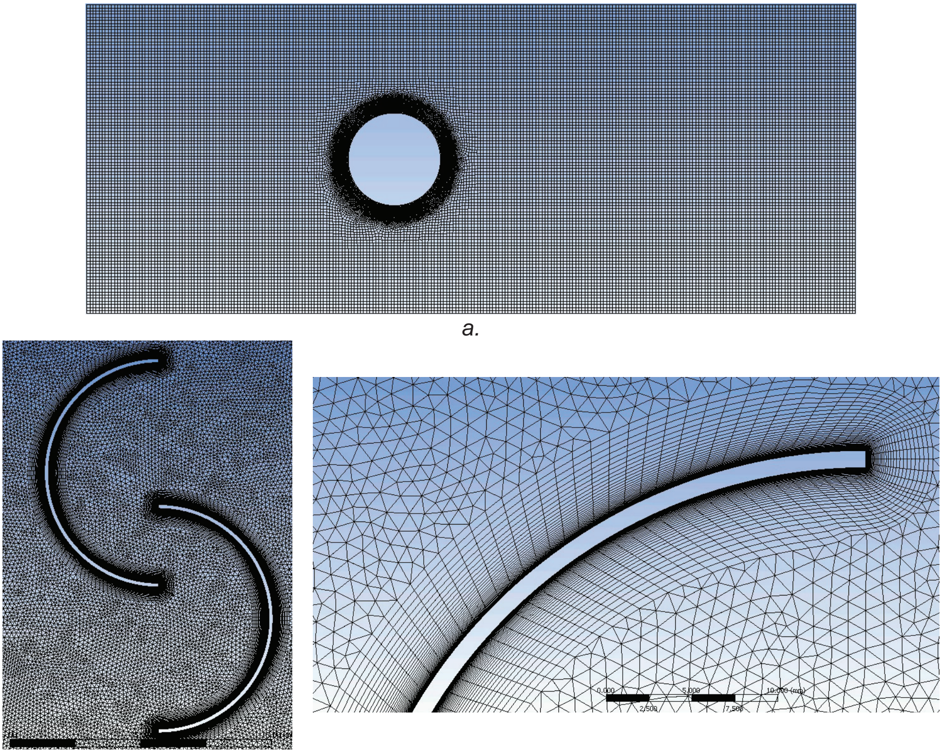
The linear (laminar sublayer) and logarithmic (turbulence sublayer) near wall laws are combined in a single one which gives the shape of the velocity profile of the first row of computational cells no matter the value of the $\boldsymbol{y^+}$ criteria.

Solver Setup

The Navier-Stokes partial differential equations for incompressible fluid flow are appropriate for modeling the operation of a Savonius VAWT, due to the fact that the flow velocity in the rotor region do not exceeds 0.3 Mach. The operation of the turbine is characterized wit high flow unsteadiness due to which the unsteady form of the governing equations is used. The system of discretized Navier-Stokes momentum equations and the continuity equation [4] are solved with the segregate scheme SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations-Corrected).

For determining the share stresses the turbulence model $k-\omega\ SST$ (Shear Stress Transition) is used. This model is consisted by two equations and combines the advantages of the $k-\varepsilon$ model for the main flow modeling and the $k-\omega$ model for the good boundary layer modeling [6, 7]. Using this turbulence model Abraham et al. [3] carried out a two dimensional modeling of a Savonius rotor operation. When comparing the theoretical and experimental results, they concluded that in the theoretical characteristic the form of the curve is well reproduced but the values were increased. In the three dimensional study carried out by Plourde et al. [8] with the $k-\omega SST$ turbulence model it can be seen a very good agreement between the theoretical and experimental results.

The value of the chosen time step corresponds to the time for which the turbine changes its angular position with $\Delta\theta = 1^{\circ}$. The results from the modelling are saved on every tenth time step in order to avoid large amounts of data. The number of inner iterations for each time step is set to 100. This allows the solution to converge when the residues of the calculated variables reaches values of the order 10^{-4}. For all simulated cases the turbulence intensity is set to be 10%. The investigation of the wind turbine is carried out for 14 different operational regimes.



a.

b. *c.*

Fig. 4 Computational Mesh

a - quadrilateral unstructured mesh in the outer stationary domain; b - triangular unstructured mesh in the area close to the blades; c - structured qudrilateral mesh in the area of the boundary layer on the blade surface

Numerical Results

Solution independence investigation from the density of the mesh is carried out. For that purpose three computational meshes with different densities are created. Each one of them has the same settings for the mesh in the boundary layer of the blades. All of the meshes are investigated at the same operational regime with tip speed ratio $\lambda = 0.375$ and undisturbed wind velocity of $\vartheta_\infty = 20 m/s$.

The Reynolds number for a flow through a wind rotor is given by:

$$\mathrm{Re}_D = \frac{\rho D u}{\mu} \tag{2}$$

where $u = \omega R$ is the peripheral velocity of the rotor, ω is the angular velocity of the turbine, D is the rotor diameter, ρ is the air density, μ is the air kinematic viscosity.

The high of the first cell row specifically selected to ensure values of the near wall criteria of $y^+ < 2.5$ for all operational regimes. The values for the Reynolds number, average and maximum values for the y^+ criteria for all investigated regimes are given in table 3.

The values of the y^+ criteria are obtained after processing the mesh data for blade 1 at four different angular positions $\theta = 0^\circ, 90^\circ, 180^\circ, 270^\circ$.

Fig. 5 show the comparison between the torques generated from the Savonius rotor obtained from three computational meshes with densities: Mesh 1 – 16750 cells; Mesh 2 – 111550 cells; Mesh 3 182100 cells. The results are showing that the torque obtained from the modeling with Mesh 2 is matching with the torque obtained from Mesh 3. Therefore the solution independence is achieved with Mesh 2. From here on Mesh 2 is used for all the simulations.

Also a solution independence study from the number of rotor revolutions is carried out. Fig. 6 depicts the torque changes for six full revolutions of the turbine. The chart is showing that periodicity in the solution is achieved after the fifth revolution. Therefore all the simulations are carried out for six full revolutions. All of the presented data is obtained from the last revolution.

Table 3

λ	Re_D	Y^+ AVE	Y^+ MAX
0.025	3422	0.74	1.79
0.0625	8557	0.78	1.64
0.125	17114	0.87	1.96
0.25	34229	0.81	1.78
0.375	51344	0.72	2
0.5	68458	0.78	2.3
0.625	85573	0.75	2.2
0.75	102688	0.75	2.2
0.875	119802	0.76	1.74
1	136917	0.73	1.71
1.125	154032	0.73	1.74
1.25	171146	0.81	1.61
1.5	205376	0.78	1.83
2	273834	0.78	1.52

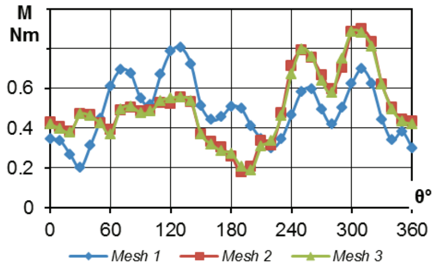


Fig. 5 Solution independence from the mesh density

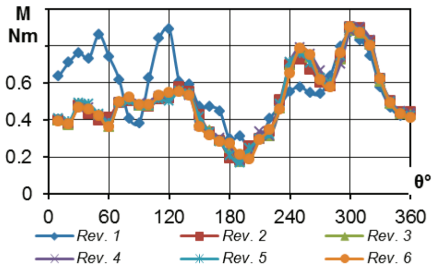


Fig. 6 Solution independence from the number of rotor revolutions

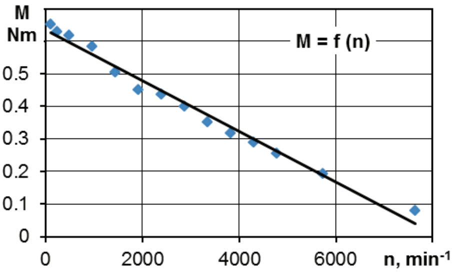


Fig. 7 Rotor torque against the rotational velocity

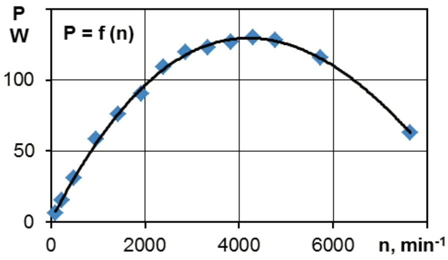


Fig. 8 Rotor output power against rotational velocity

The results for the average values of the torque against the rotational velocity are shown on fig. 7. As can be seen from the graphic the maximum torque value is achieved in the area of the lowest rpm's. Its maximum value is $M \approx 0.7\,Nm$ reached at $n \approx 240\,\text{min}^{-1}$. With the increase of the rotational velocity the torque is decreasing until it reaches values around zero at $n \approx 7640\,\text{min}^{-1}$. On fig. 8 presents the turbine output power against the rotational velocity. The output power is obtained by:

$$P = M\omega\,. \tag{3}$$

The maximum value of the output power is $P \approx 150W$ achieved at $n \approx 4800\,\text{min}^{-1}$.

Conclusions

The presented methodology for numerical, two dimensional modeling the operation of a VAWT uses the technique Sliding Mesh. The size of the computational domain is selected according to recommendations from the reference literature.

The values for the y^+ criteria at all operational regimes do not exceed 2.5, which ensure the modeling of the boundary laminar sublayer.

A mesh independence study is carried out, through which the mesh with the optimal density is evaluated.

From the solution independence study from the number of rotor revolutions is found that the periodicity in the solution is achieved after the fifth revolution.

Теоретичните резултати ясно показват адекватността на метода да моделира работата на вятърните турбини с вертикална ос. Предложената методика е и адекватен инструмент за получаване на теоретичните им характеристики.

The theoretical results clearly show the adequacy of this approach to successfully model the operation of VAWT. The proposed methodology is also an adequate tool for obtaining their theoretical characteristics.

Reference

[1] Ахмедов А., Кр. Тужаров, Г. Попов, И. Желева, И. Николаев; Кл. Климентов. Теоретични модели за изследване на ветротурбини с вертикална ос на въртене тип Дариус – моментни модели. Механика на машините, 2013, брой 2, стр. 27-32, ISSN 0861-9727.

[2] Ахмедов А, Кр. Тужаров, Г. Попов, И. Желева, Кл. Климентов, И. Николаев. Вихров и каскаден модели за изследване на ветротурбини с вертикална ос на въртене тип Дариус. В: Научни трудове на РУ "А. Кънчев", том 51, сер. 1.2, Русе 2012, Русе, 2012.

[3] Abraham J P, Mowry G S, Plourde B P, Sparrow E M, Minkowycz W J. Numerical simulation of fluid flow around a vertical-axis turbine. Journal of Renewable and Sustainable Energy 2011; 3 (3): 1–13.

[4] ANSYS Fluent 14.0, User Guide.

[5] Cummings, R.M., Forsythe, J.R., Morton, S.A., Squires, K.D., Computational Challenges in High Angle of Attack Flow Prediction, 2003, Progr. Aerosp. Sci. 39(5):369-384.

[6] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 1994; 32(8): 1598–605.

[7] Menter F R. Zonal two-equation k–ω turbulence models for aerodynamic flows. In Proceedings of the 24th AIAA Fluid Dynamics Conference, Orlando, FL; 6–9 July 1993.

[8] Plourde B D, Abraham J P, Mowry G S. Simulations of three-dimensional vertical axis turbines for communication applications. Wind Engineering 2012;36 (4): 443–54.

[9] [9] Simao Ferreira, C. J., Bijl, H., van Bussel, G., van Kuik, G.: Simulating Dynamic Stall in a 2D VAWT: Modeling Strategy, Verification and Validation with Particle

Image Velocimetry Data, The Science of Making Torque from Wind, Journal of Physics: Conference Series 75, 2007.

[10] Simao Ferreira, C.J., van Bussel, G., Scarano, F., van Kuik, G.: 2D PIV Visualization of Dynamic Stall on a Vertical Axis Wind Turbine, AIAA, 2007;

Corresponding author:

Ahmed Ahmedov MSc, Department of Thermotechnics, Hydraulics and Ecology, University of Ruse Angel Kanchev, e-mail: aahmedov@uni-ruse.bg

The report is reviewed.