Numerical analysis of the flow around the Bach-type Savonius wind turbine

K Kacprzak, K Sobczak
Institute of Turbomachinery, Lodz University of Technology, 219/223 Wólczańska Street, 90-924 Łódź, Poland
E-mail: kokacprzak@gmail.com

Abstract. The performance of the Bach-type Savonius wind turbine with a constant cross-section is examined by means of quasi 2D and 3D flow predictions obtained from ANSYS CFX. Simulations were performed in a way allowing for a comparison with the wind tunnel data presented by Kamoji et al. The comparison with the experiment has revealed that 2D solutions give much higher deviation from the reference data than the 3D ones, which guarantees a good solution quality. It can be stated that even simplified (lack of laminar-turbulence transition modelling and a coarser mesh) 3D simulations can yield more accurate results than complex 2D solutions for turbines with a low aspect ratio. The paper also presents a systematic analysis of the most characteristic flow structures which are identified in the rotor.

1. Introduction
In recent years, investigations in the field of wind energy have been considerably intensified due to the worldwide energy crisis, high global CO$_2$ emission and grid overload. However, there is still a great potential for further development of reliable distributed energy converters for moderate wind speed conditions. Real-scale or model experiments involve huge costs and hazards. On the other hand, computational studies make the whole process much faster, however, in this case, valid numerical models of flow phenomena are the key aspect to obtain realistic results.

The Savonius wind turbine is a vertical axis wind turbine (VAWT) that was proposed by Savonius in the 1920s. It has a characteristic S-shape of the rotor composed of two separated or connected scoops. A typical power output of the Savonius turbine classifies it as a small wind turbine. Decentralised power generation is now one of the fastest-growing forms of providing electricity for domestic and residential needs and this turbine can be an attractive investment for homeowners and small business. Hence, there has been increased interest in studies of the Savonius rotor, and significant efforts have been made to improve its aerodynamic performance. In particular, there has been a need for better understanding of the unsteady flow around the Savonius rotor.

This increased interest in the Savonius wind turbine results from increasing significance of the urbanised areas, which address specific demands. The Savonius rotor seems to satisfy these particular expectations. Its main advantages can be described as follows [1,2]:

- wind direction independence,
- ability to operate in a wide range of wind conditions,
• no complex control mechanism is required,
• structure can be placed on the ground,
• low noise emission,
• high starting torque.

All the above features combined with a compact size of this kind of turbines make them suitable for the residential use. However, Savonius turbines are not free of drawbacks like relatively small efficiency or low rotational speed [3,4]. Therefore, constant search for better designs, in order to improve the performance of the rotor is required. Current development in Computational Fluid Dynamics (CFD) makes it possible to conduct a comprehensive study of different designs of the Savonius rotor without committing a significant amounts of time and money for experimental investigations.

2. Studied geometry
In this paper the Bach-type rotor was examined by means of numerical simulations (figure 1). In order to investigate the impact of three-dimensional effects on the overall performance of the turbine full scale 3D simulations were carried out together with a series of 2D simulations. Dimensions of a numerical model directly correspond to a low aspect ratio turbine examined experimentally by Kamoji et al. [5]. In this way, it was possible to verify obtained results and check the influence of different task configurations (2D simplification, mesh refinement). Simulations are conducted for a rotor with height \( H = 0.154 \) m; diameter \( D = 0.2 \) m; without overlap - ratio \( e/D = 0.0 \); aspect ratio \( H/D = 0.77 \); blade arc angle \( \psi = 135^\circ \); blade shape factor \( p/q = 0.2 \); end plate parameter \( Do/D = 1.33 \) (figure 2).

![Figure 1](image1.png)  
**Figure 1.** Model of investigated Bach-type turbine (due to its symmetry only a half of turbine is presented).

![Figure 2](image2.png)  
**Figure 2.** Geometrical parameters of the Bach-type turbine.

3. Simulation definition
In this study both 2D and 3D simulations were carried out with ANSYS CFX. For the purposes of this investigations the model was composed of two domains (Fig 3.). The first one (rotating) comprised the wind turbine blades and endplates and the second (stationary), simulated the wind tunnel with dimensions in accordance with the experimental test stand data [5]. The rotating domain radius was twice as big as the rotor radius. Such a domain size was chosen in order to assure that vortex structures in a close vicinity of the rotor are solved in a rotating frame of reference and it also minimised numerical errors due to numerous transitions of fluid through the domain interface. Due to the rotor symmetry, only a half of the turbine was solved. Simulation definition in the case of 2D model is
similar, however, due to a nature of the ANSYS CFX solver, a slice (1 mm thickness) of 3D domain near the symmetry plane was used. Meshes for the given geometry were prepared using Meshing software from ANSYS Workbench. In the performed simulations a high-quality, unstructured, hexahedral mesh was utilised. Described grid was comprised of hexahedral layers, perpendicular to the rotor axis. Throughout the analysis different mesh densities were employed. First of all, a coarse mesh (~6·10^6 nodes) was used, where wall functions were utilised to bridge the viscosity-affected region between the wall and the fully-turbulent region. In case of the fine mesh (~15·10^6 nodes) y⁺ values for the buckets were lower than 2, which made it possible to fully resolve the boundary layer, which due to considered flow structure is of a very high relevance.

![Figure 3. Orthogonal projection of simulation domains.](image)

Taking into account low wind speed in the described case, density changes were relatively low, that is why, it was assumed that air in this case is incompressible. Due to constant changes of the blade position in reference to the wind direction, the flow structure is very complex and extensive separations occur. Hence, the unsteady Reynolds Averaged Navier-Stokes (RANS) simulations and especially turbulence modelling in this case is a challenge. In the described simulations Shear Stress Transport (SST) turbulence model was utilised. It was successfully introduced in similar studies [6,7]. The selection of this two-equation model was based on the fact that it was designed to give accurate prediction of the flow separation regions under the adverse pressure, when an adequate mesh refinement (as in this study) is applied. It is achieved by an implementation of the transport effects to the formulation of the eddy viscosity. That is why, it is used whenever a high boundary layer accuracy is required [8], which is a case in this investigation. Utilising this turbulence model also enabled the laminar-turbulent (LT) transition modelling. The γ-θ model was applied, which was demonstrated by numerical studies [9] to have an influence on the results agreement with the experimental data. This approach required mesh refinement to obtain y⁺ < 2. This condition was fulfilled for the buckets. In case of endplates, due to the fact that the LT transition is not of a high importance in this region and taking into account the calculation time, y⁺ values were higher.

At the inlet to the computational domain a wind velocity (9 m/s) boundary condition was imposed. This velocity corresponds to Re = 120 000. Additionally, low turbulence intensity (1%) was set as in [5]. At the outlet atmospheric pressure was imposed (1 bar). Due to the fact that the turbine as well as the expected flow structure have a symmetry plane in the middle of the turbine height, in 3D simulations a symmetry boundary condition was set on this plane. In case of 2D calculations symmetry was imposed on both sides of the modelled layer which is typical approach in 2D simulations in ANSYS CFX. For turbine blades and endplates no-slip wall boundary condition was
assumed (zero velocity on the wall surface). At sides and bottom of the domain (away from the rotor) the free-slip wall condition was imposed. Adjusting the rotational velocity of the rotor made it possible to investigate different tip speed ratios.

\[ TSR = \frac{\omega \cdot R}{U} \]  

where \( \omega \) – turbine angular velocity, \( R \) – turbine radius, \( U \) – wind speed.

In order to achieve conformity between mesh cells on two sides of the domain interface for the whole simulation, timestep for every rotational velocity was adjusted to achieve precisely 360 timesteps per one revolution.

ANSYS CFX uses an element-based finite volume method [8]. Second order advection (high resolution) scheme were utilised. Second order Backward Euler method was used for discretization of time terms. In order to properly control convergence, the level of the residual target was set to \( 10^{-5} \), with a number of iterations for each timestep (so called internal convergence loops) between 3-10. In practice 6 iterations and the root mean square Courant number equal to 3 were not exceeded. That implies that the timestep is small enough to adequately resolve unsteady structures. It has to be mentioned that the results after 6-8 revolutions of the rotor yielded repeatable cycles, so only the data beyond the eighth revolution was taken into account during the postprocessing of the solution results.

4. Simulation results

In order to verify the results of conducted simulations, obtained data were confronted with the experiment results from Kamoji et al. [5]. Relation between coefficient of power \( c_p \) (ratio of the rotor and wind power ) and tip speed ratio for Bach-type turbine investigated by Kamoji et al. is presented in figure 4 together with 2D and 3D simulation results.

\[ c_p = \frac{2T \cdot \omega}{\rho \cdot U^3 \cdot D \cdot H} \]  

where \( T \) – torque, \( \rho \) – air density.

![Figure 4](image-url)  

**Figure 4.** Comparison of the \( c_p \) coefficient from simulations and experiment.
It can be noticed that there are certain discrepancies between the experimental and simulation results. In the case of 2D results, it has to be underlined that the 2D model does not take into account all the three-dimensional effects (especially vortices, which are the source of major dissipations) on the blades in the vicinity of the endplate. Moreover, additional losses may occur due to turbine shaft rotational resistance. However, in the case of 2D simulations for TSRs below 1.0 obtained power is underestimated, which can result from imperfections of the used method e.g. turbulence modelling in 2D. On the other hand, 3D results for coarse mesh are unambiguous and indicate small (a few %) overprediction of $c_p$ relatively to the experimental data. Application on the laminar-turbulent (LT) transition model in case of the 3D fine mesh improved the estimation of power coefficient. This can be due to better separation prediction. For example delayed separation of the boundary layer reduces the pressure drag on the returning bucket, which produces higher lift force participation on the resultant force, increasing the torque of the rotor.

![Figure 5. Comparison of the $c_p$ coefficient from simulations and experiment for TSR=0.8.](image)

5. Flow structure analysis

In order to assess which parameters assure the most accurate solution, simulations for multiple task configurations were performed. All of them were conducted with TSR = 0.8 as the most representative case (the highest power output). Simulations were done for 2D and 3D cases, for coarse and fine meshes, without and with the laminar-turbulent (LT) transition modelling. Obtained values of the pressure coefficient are presented in figure 5. One can see that the LT transition modelling have serious influence on the 2D solution. In the case of the fully turbulent boundary layer data are highly overpredicted (more than 20%), whereas for the case with LT modelling, it is underdetermined (8.5%). 3D simulations give more straightforward results. There is small difference between the solutions of the fully turbulent boundary layer for the coarse and fine meshes (overdetermined by approx. 8.5%). Much better agreement with the experimental data is obtained if the LT transition is taken into account. In this case the difference is 4%. It seems that for a low aspect ratio rotor simpler 3D simulations (coarse mesh, lack of the LT transition modelling) can yield more accurate results than more complex 2D solutions.

In order to examine the flow structure for different rotor positions, the simulation concerning TSR = 0.8 was chosen as the most representative. It is due to the fact that the coefficient of power attains the highest value for this TSR. The variation of $c_p$ for one, full revolution of the rotor is presented in figure 6. The most characteristic points, representing $c_p$ extremes, are marked. A and C refer to the maximum values, while B and D to the minimum ones.
It can be observed, that the position of the rotor corresponding to the highest torque is attained twice throughout the revolution for blade positions almost "aligned" with the flow, so it can be concluded that it consists of two almost equal cycles. It is also visible that the coefficient of power drops to almost 0 for the blade positions "normal" to the incoming wind.

![Power Coefficient Graph](image)

**Figure 6.** Coefficient of power for the Bach-type wind turbine for TSR=0.8.

In figure 7 one can see that the flow structure is very complex and time dependent. In the angular position A, when the highest torque is attained, a clear fluid acceleration on the convex side of the advancing blade is observed (moving in agreement with the wind direction) with subsequent separation of the boundary layer. This acceleration results in production of the lift force, which drives the turbine. This basic phenomenon coexists with numerous recirculation zones at the convex side of the advancing blade and both sides of the returning blade. In the case of the angular position B, the stagnation at the returning blade balance the positive torque due to wind action on the concave side of the advancing blade. Flow structure obtained in this numerical study directly corresponds to other research, where the flow in the Bach-type Savonius wind turbine was studied by means of PIV [10].

![Velocity Distribution](image)

**Figure 7.** Distribution of velocity in the stationary frame for the characteristic points A and B at the symmetry plane.
In order to observe the flow structure from a different perspective figure 8 was prepared. It shows streamlines for angular positions of the rotor: 0, 45, 90 and 135°. Similarly as in figure 7, a high level of flow structure complexity with several recirculation zones is observed. It can be noticed that they have 3D character, which may explain why 2D simulations yielded poorer agreement with the experiment.

Figure 8. Streamlines in stn. frame of reference for different angular positions of the Savonius rotor.

Four most dominant structures are depicted with different colours. The first one, marked in red, is an advancing blade tip vortex, which is created at the upper tip of the advancing blade and develops downstream to much bigger size. It firstly occurs at the angular position of 90°. At angular positions of 135° and 180° (0°) it grows and separates from the tip of the blade at 225° (45°). Another structure, which is marked in black is an advancing blade boundary layer separation, which forms close to the axis of the turbine. This structure originates around an angular position of 0°. Its another instance
from the previous period (half of the rotation) forms concave side recirculation and it is marked in
orange. They are formed due to the fact that the Coanda effect can no longer keep the flow attached to
the blade and it slowly separates. The last dominant structure that may be observed (marked in purple)
is a result of multiple structure interactions and the part of the orange structure detachment from the
returning blade. Together with other detached structures it creates one complex conglomerate
downstream of the rotor. It may be noticed that in particular all the structures detaching from the
turbine blades exhibit predominantly complicated flow character significantly modified close to the
endplate due to its boundary layer interaction.

Blade configuration of the Bach-type Savonius rotor, including reduced blade arc angle (in
comparison to a typical Savonius turbine) creates higher lift force on the advancing blade for the
angular position of the rotor corresponding to points A and C. At the same time it results in lower
resistance while spinning in the air. Another crucial factor affecting the turbine flow field is a lack of
an overlap distance, which would diminish the negative torque on the returning blade.

6. Conclusions
On the basis of the presented data the following conclusions can be drawn:

- Comparison of experimental data and obtained simulation results clearly show that 2D
  simulations exhibit higher deviation from the experimental results in comparison to 3D ones.
- 3D simulations behave in more predictable and stable manner and are less sensitive to mesh
density and changes in the simulation definition.
- In case of the Bach-type turbine for a low aspect ratio, flow structure is too complicated to be
  properly modelled by means of 2D simulations, therefore even a simplified 3D solution yields
better results than complex 2D one.
- Flow acceleration on the advancing blade aligned with the flow is one of the key effects
  contributing to the torque of the rotor.
- Blades in the normal to the flow position produce minimal torque.

7. References
[1] Spera D 2009 Wind Turbine Technology: Fundamental Concepts of Wind Turbine Engineering
(ASME Press)
[2] Wagner H J and Mathur J 2009 Introduction To Wind Energy Systems: Basics, Technology and
Operation. Green Energy and Technology (Springer)
[3] Fujisawa N and Gotoh F 1994 Experimental Study on The Aerodynamic Performance of
a Savonius Rotor J. of Sol. En. Eng. pp 148-152
[4] Ogawa T 1984 Theoretical Study on The Flow About Savonius Rotor ASME J. Fluids Eng. pp
85-90
[5] Kamoji L, Kedare S and Prabhu S 2009 Experimental investigations on single stage modified
Savonius rotor App. En. pp 1064-1073
[6] Zhao Z, Zheng Y and Xu X 2009 Research on the improvement of the performance of Savonius
rotor based on numerical study Sus. Pow. Gen. and Supp. pp 1-6
[7] Zhou T and Rempfer D 2013 Numerical study of detailed flow field and performance of
Savonius wind turbines Ren. En. pp 373-381
[8] ANSYS CFX v14 Help
[9] Kacprzak K, Liśkiewicz G and Sobczak K 2013 Numerical investigation of conventional and
modified Savonius wind turbines Ren. En. pp 578-585
[10] Dobreva I, Massouha F 2011 CFD and PIV investigation of unsteady flow through Savonius
wind turbine En. Proc. pp 711-720

8. Acknowledgments
This article was funded from the public budget by the Ministry of Science and Higher Education as the
“Diamond Grant” research project no. DI2011 000441.