Technical Sciences
Academy of Romania
www.jesi.astr.ro

Journal of Engineering Sciences and Innovation
Volume 4, Issue 2 / 2019, pp. 145 - 154

A. Mechanical Engineering

Received ***18 February 2019*** *Accepted* ***30 May 2019***
Received in revised form ***30 April 2019***

Numerical simulation of transient aerodynamic processes in the vertical axis wind turbine rotor

BOSTAN VIOREL, GUȚU MARIN*, RABEI ION

Technical University of Moldova, Bd. Ştefan cel Mare, 168, MD-2004, Chişinău, Republic of Moldova

Abstract. This paper describes the steps of elaboration of calculation model for dynamic simulation of a small vertical axis wind turbine rotor. The calculation model is based on the finite element analysis ANSYS CFX software. The CFD model is used to determine the performance of the wind turbine rotor for different settings (tip speed ratio, solidity, mesh size etc.). The verification of the proposed model is done by hand calculation of the rotor performance and comparison of the real wind turbines output.

Keywords: vertical axis wind turbine, finite element analysis, transient aerodynamic interaction, CFD model.

1. Introduction

An important tool in engineering that generally lead to cost and time savings during product development are simulations [1,2]. Finite element analysis serves as a base for the present work. Besides choosing the appropriate mathematical model behind physics of the simulated system, it is important to choose the right shape and size of the finite elements. It is also important that the elements are well adapted for the specific system to be analyzed.

This work presents a CFD model created for determining the performances of a vertical axis wind turbine. The CFD model is made using ANSYS CFX software. The CFD model is used to determine the power curve of a 4 kW modeled wind turbine. For this power there are wind turbines developed by several companies. Comparison of the simulated wind turbine power output with real wind turbines power output is considered as validation of the CFD model.

* Correspondence address: marin.gutu@pmai.utm.md

2. Rotor's geometry and fluid domain modeling and meshing

The figure 1 presents the geometrical model of the 4 kW vertical axis wind turbine rotor, used for the later simulations, made using the software SolidWorks. The rotor dimensions were determined using well known relation:

$$P := \frac{\rho \cdot A}{2} \cdot cp \cdot V^3 \tag{1}$$

where ρ - air density;
A – rotor swept area $A = H_r \times D_r$;
V – wind speed (calculation wind speed is 11 m/s);
c_p – power coefficient ($c_p = 0.3$).

The chord of the turbine blade is 0.3 m, though more chord lengths were analyzed. The helical angle of the blades is 60°.
The rotor geometry, designed using SolidWorks, was then imported into the ANSYS DesignModeler software.
The dimensions of the computational fluid domain were chosen taking into account the recommendations of [3] so that the boundaries of the field do not influence the free flow of the air. The simulated fluid domain was divided into two subdomains: the Stator (static) subdomain and the Rotor subdomain inside of it (of cylindrical form, which rotates around its axis). Figure 3 shows the considered fluid domains.
The mesh used for finite element analysis of the rotor was generated using the ANSYS Meshing Workbench. This is integrated software that offers various meshing strategies.

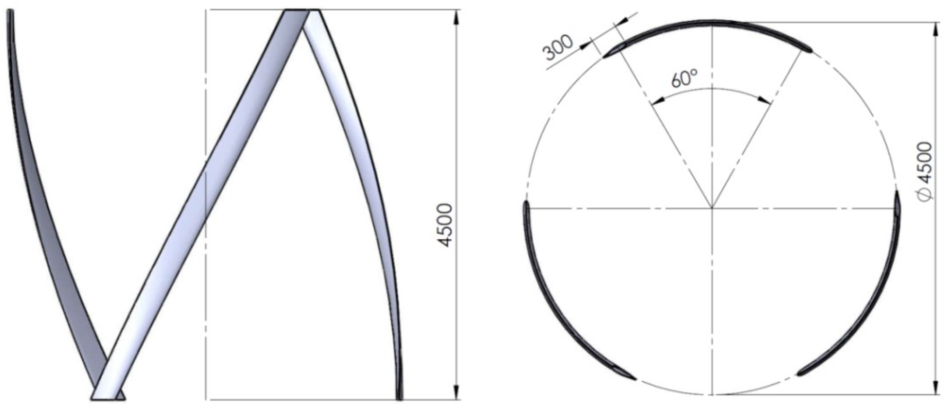
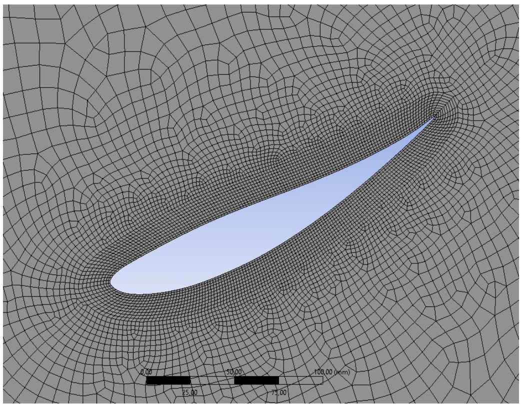


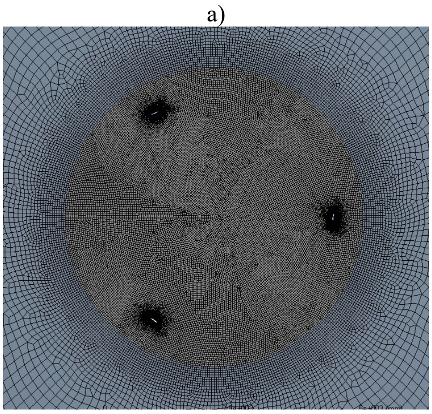
Fig. 1. Rotor geometry.

After importing the geometric model the following regions were defined [4]: (Inlet – the face with black arrows on the perimeter), (Outlet – the opposed face of the Inlet), (Openings – the four side faces), and the common regions between Stator and Rotor (Fluid-Fluid). The basic dimensions of the mesh are as follow: the minimal size of the inflation around the blade = 0.7 mm and the maximum size of the side of one face = 220 mm (fig. 2).

The transition from the fine-meshed areas to the gross meshed ones was done by specifying the Growth Rate = 1.1 expansion factor. The maximum variation of the characteristic dimensions of two adjacent elements is not bigger than 5%. The entire domain was meshed into approx. 4 200 000 finite elements.



a)



b)

Fig. 2. Mesh of the rotor fluid domain (a) and details of boundary layer around the blade (b).

The effects formed on the blades' surfaces are very important because here it is where the lift and drag are formed, boundary layer separation occurs and other important effects take place In order to obtain more accurate results in the close proximity of the blades' faces where the boundary layer forms, rectangular finite elements have been generated by expanding them from the surface of the blade outwards.. This was done using Inflation Layer technique around blades' surfaces: Number of Layers = 9, the Growth Rate = 1.18 (relative thickness between two adjacent layers), and Growth Rate Type = Geometric. Figure 2, (b) shows the mesh details around the blade.

3. Boundary conditions setup

Problem setup was done using CFD module of the ANSYS software. In order to verify the conversion efficiency of the turbine, several modes have been simulated. There were considered two different airfoils with different chord lengths and each chord length is simulated under different tip speed ratios. The wind speed considered for simulations is 11 m/s. The parameters of interest that were analyzed are presented in the table 1.
As option for analysis type the Transient Blade Row model is chosen. The boundary conditions imposed are the following: entry into the computing field is made by the boundary determined by the rectangular base of the upstream Stator fluid domain [5].

Table 1. Analyzed rotor parameters

Airfoils	Chord length, m	Wind speed, m/s	Tip speed ratios	Rotational speed, min^{-1}
NACA 0018	0.2; 0.3; 0.4	11	3	126
			3.5	147
Wortman FX 63 137			4	168
			4.5	189

At this border were imposed Inlet boundary conditions with the specification of the uniform and constant velocity distribution in the fixed reference system (V_0, 0, 0), where V_0 is wind speed. Outlet from the computing domain is made by the downstream rectangular base by specifying the Outlet boundary conditions with the average static pressure 0 Pa. Surfaces of the Stator fluid domain are subjected to Walls boundary conditions with the free-slip specification that simulates a zero-adhesion virtual wall. Blades surfaces are subject to Walls boundary conditions with no slip specification which does not allow mass or energy transfer, and the speed on these surfaces is considered equal to 0 in relation to the speed of the adjacent cells. The surfaces at the intersection of the two Stator and Rotor subdomains are interface surfaces that model the connection of the two subdomains through the GGI method. Rotor rotation simulations specify Domain motion - Rotating and indicate the angular velocity of relative rotation ω. At this stage extra attention is required to certain details such as the direction of rotation of the rotor and wind direction, which can be changed with the (-) sign. In order to draw wind turbine rotor power curve several operating modes have been simulated, (table 2). Figure 3 shows calculation domain.

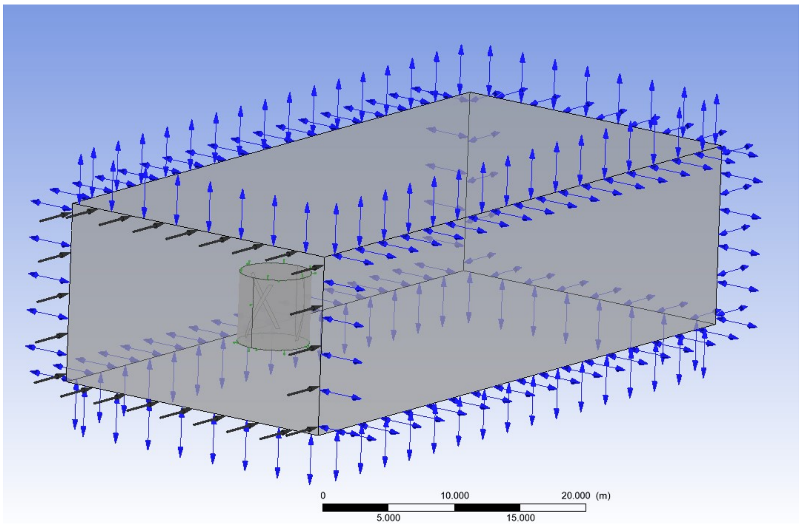
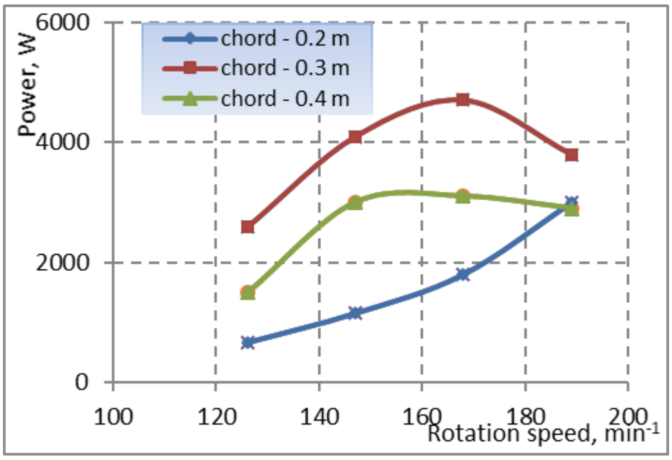


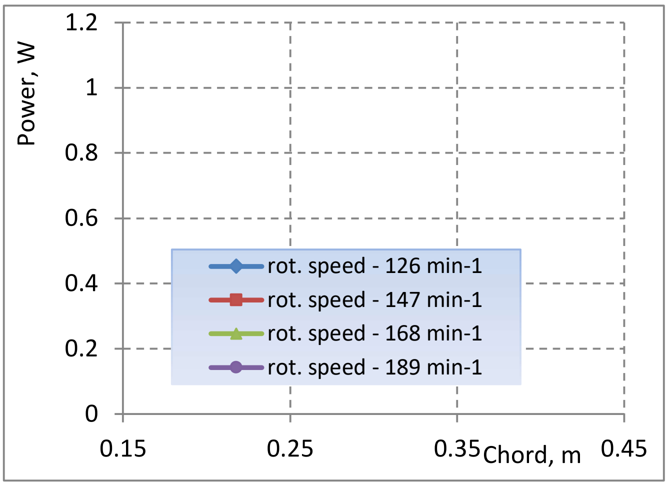
Fig. 3. The fluid domains.

4. Computational fluid dinamics results

Due to the fact that the preliminary results of the numerical analysis of the rotor performance are better for NACA 0018 airfoil, this airfoil has been selected for subsequent simulations.
The solving of discretized equations was performed in parallel using all 16 logical cores. The goal is to obtain 4 – 4,5 kW of power at a wind speed of 11 m/s. First of all was determined optimal airfoil chord length. The variable of interest was set the power output.
The simulations were carried out and the results are presented in figure 4. We can conclude that the optimal chord length for this rotor is 0.3 m.



a)



b)

Fig. 4. The parameters of the 4 kW rotor at the wind speed of 11 m/s for NACA 0018 (*a*), the dependence of the rotor's power output on the airfoil chord for different angular velocities (*b*).

Next step is to determine the power curve of the rotor, that is to determine by simulation the power output of the rotor equipped with the blades of 0.3 m chord length at each wind speed

The results of the simulations are synthetized in the figure 6. It can be noticed from the figure that this turbine is efficient at high wind speeds and not very efficient at low wind speeds, as for a wind speed of 6 m/s the power output is approximately 500 W.

In order to check flow field around the rotor, the velocity contour section for 11 m/s wind speed is depicted in the figure 5. From the presented results it appears that the wind turbine three-dimensional blade has a performance reduction linked to vortex diffusion after flow separation; this is in accordance with the literature.

For a more detailed presentation of this effect, the vortex core regions are shown in figure 7. Three different time steps were considered. Also the optimal rotational speed was considered for each rotor, which is 107 rpm for FX 63-137 and 122 rpm for NACA 0018. It appears that as the rotational speed increases, vortex core regions extend more and more, influencing the upcoming blade. This phenomenon is more evident in the blade based on FX 63-137 airfoil, where vortexes accumulate mainly because of not ensuring an optimal operating speed. The vortices are evenly distributed along the blade trailing edges. The cause of this is that this airfoil is designed for low wind speeds.

For the validation of the results the characteristics of the wind turbines Urban Green Energy Vision Air 4 kW and Quiet Revolution 5 kW were analyzed. The comparison parameters are presented in the table 2.

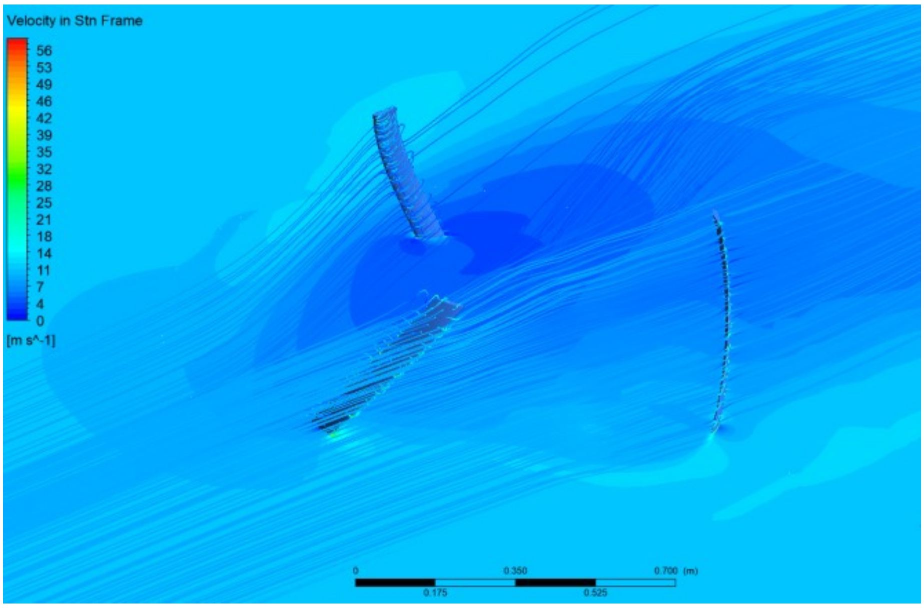


Fig. 5. Velocity contour for 11 m/s wind speed.

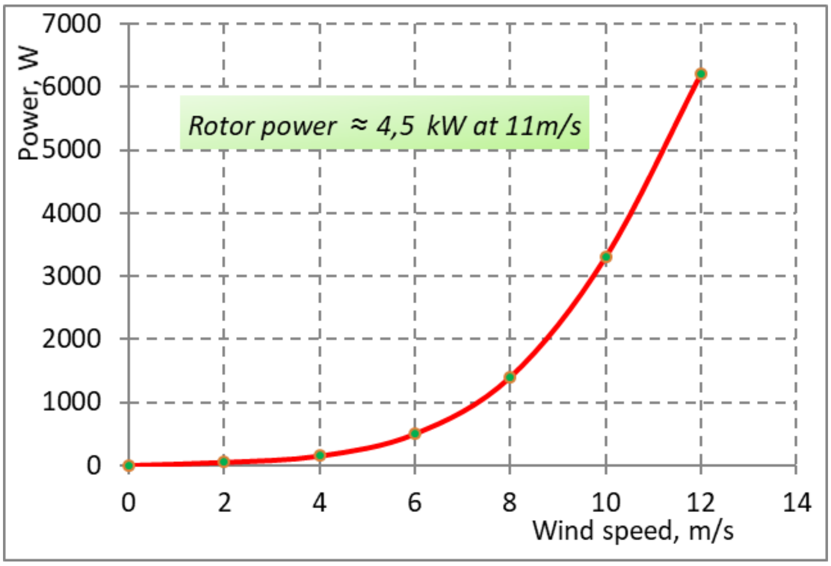
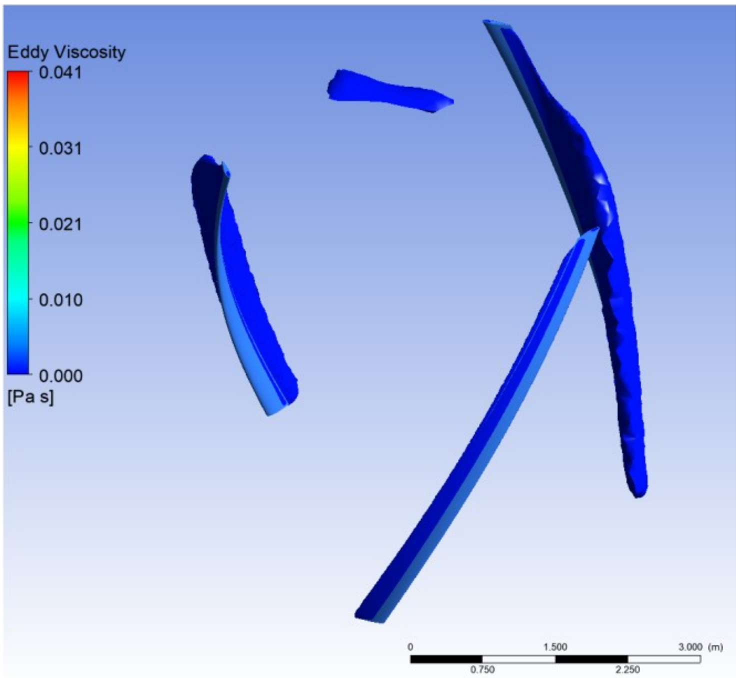
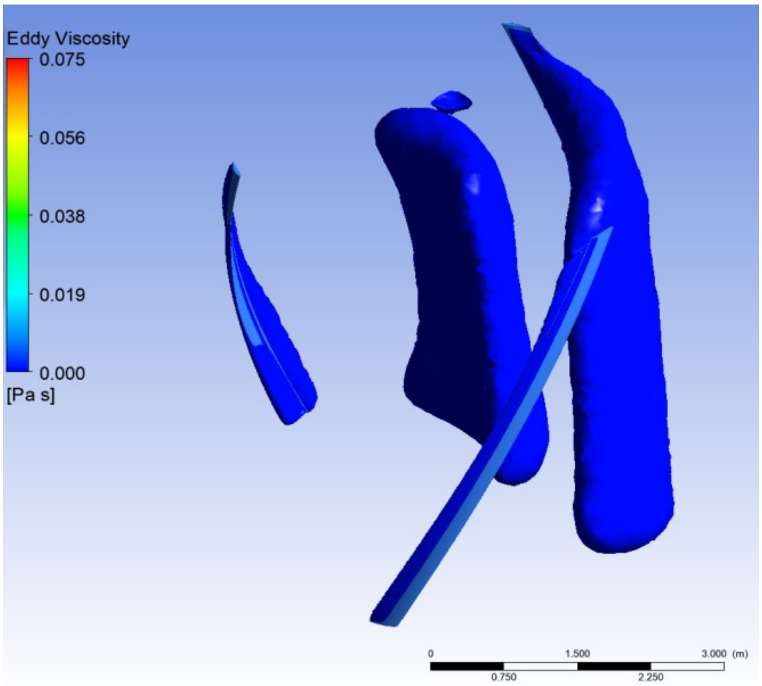


Fig. 6. Power curve.



a)

Fig. 7. Tip vortex diffusion at 8 m/s wind speed: *a* - NACA 0018 optimal rotation speed *n* = 122 min^{-1}



b)

Fig. 7. Tip vortex diffusion at 8 m/s wind speed: *b* - FX 63-137 optimal rotation speed *n* = 107 min^{-1}

Table 2. Comparison of wind turbines parameters and simulated rotor.

Wind turbine	Simulated rotor	Quiet Revolution [6]	Urban Green Energy Vision Air 5 [7]
Rated power	4,5 kW	4,5 kW	1,5 kW
Wind Speed	11 m/s	11 m/s	11 m/s
Rotation Speed	165 min^{-1}	≈ 230 min^{-1}	≈ 110 min^{-1}
Estimated tip speed ratio	$\lambda \approx 3.8$ - 4	$\lambda \approx 3.8$	$\lambda \approx 2.2 – 2.3$
Cut-in Wind Speed	4 m/s	4.5 m/s	3.6 m/s
Dimensions	H = 4.5 m_ D = 4.5 m	H=5.5 m _ D=3.1 m	H=5.2 m_ D=3.2 m
Solidity of rotor	≈ 0.22	≈ 0.18	≈ 0.34
Airfoil type	symmetrical (NACA 0018)	symmetrical	asymmetrical
Rotor shape			

5. Conclusions

For the 4 kW wind turbine (diameter 4.5 m, height 4.5 m) with blades based on NACA 0018 airfoil, the maximum power output at a speed of 11 m/s is obtained for a chord length of 0.3 m running at a rotational speed of 168 rpm, which corresponds to a tip speed ratio of 4.

The rotor based on FX 63-137 airfoil shows poor performance for the same geometrical characteristics (height, diameter and chord length) and flow conditions (wind speeds) when compared to the rotor based on NACA 0018 airfoil. Because this airfoil is designed to operate at low wind speeds, perhaps a larger solidity and poor wind conditions need to be considerate when dealing with this airfoil. These aspects are going to be the subject for further simulations.

The separation of the boundary layer for the FX 63-137 bladed rotor cause more violent vortexes when compared to NACA 0018 bladed turbine. From here a larger aerodynamic stall is experienced so the power output is lower.

Comparing the parameters of these rotors is perhaps not the most appropriate in terms of the geometry. However, the rotor swept area may be considered a parameter that indicates a good correlation of the power delivered between the rotors. The big differences are caused by the different rotor solidity (the airfoil cord length). Thus, the high-solidity rotor has a lower yield at high wind speeds. This has also been demonstrated through simulations. The airfoil type also influences the performance of the rotor. The Energy Vision Air 5 turbine rotor featuring blades with an asymmetrical airfoil indicates overall poor performance. Their advantage is the lower cut-in wind speed (2.5 vs. 4 m/s).

References

[1] Bostan I., Dulgheru V., Sobor I., Bostan V., Sochirean A., *Renewable energy conversion systems*, Bons Offices, Chișinău, 2007.

[2] Bostan I., Gheorghe A., Dulgheru V., Sobor I., Bostan V., Sochirean A., *Resilient Energy Systems, Renewables: Wind, Solar, Hydro*, Springer, VIII, 2013.

[3] Mohamed M.H., Ali A.M., Hafiz A.A., *CFD analysis for H-rotor Darrieus turbine as a low speed wind energy converter,* Engineering Science and Technology, an International Journal, Volume **18**, Issue 1, March 2015.

[4] Bostan V., *Modele matematice în inginerie, probleme de contact, modelări și simulări numerice în aero-hidrodinamică*, Chișinău: „Bons Office", 2014.

[5] Bostan I., Dulgheru V., Bostan V., Ciupercă R., *Antologia invenţiilor.* Anthology of Inventions: renewable energy conversion systems), Vol. 3, Bons Offices, Chișinău, 2009.

[6] https://www.quietrevolution.com/ (accessed on 14.05.2018)

[7] http://visionairwind.com/ (accessed on 16.05.2018)