The Analysis of Wind Turbine with Horizontal Rotation Axis with the Use of Numerical Fluid Mechanics

Damian Mazur, Mariusz Trojnar and Andrzej Smoleń

Abstract The analysis of horizontal rotation axis wind turbine of type H-rotor is presented in this article. The calculation has been made using a 3D model of horizontal rotation axis turbine for three wind flow speeds: 5, 10, 15 m/s, with the stable rotational speed of the rotor of 200 rotations per minute. The analysis of wind turbines requires building a 3D calculation model. The building of such a model is very time-consuming and the calculations involving numerical fluid mechanics require substantial calculation power. The article presents 3D CAD model of horizontal rotation axis wind turbine in ANSYS calculation environment, discrete net, and boundary conditions and the results of calculating horizontal rotation axis wind turbine with the use of numeric fluid mechanics.

Keywords Wind turbine · Horizontal rotation axis · CFD analysis · Turbine-generated power

1 Introduction

The article describes the analysis of horizontal rotation axis wind turbine of type H-rotor. The calculation has been made of horizontal rotation axis wind turbine using a 3D model of horizontal rotation axis turbine for three wind flow speeds: 5, 10, 15 m/s, with the stable rotational speed of the rotor of 200 rotations per minute.

This analysis has been made to illustrate and follow the turbine behavior and to present the calculation opportunities of horizontal rotation axis turbines.

D. Mazur (✉) · M. Trojnar · A. Smoleń
Department of Electrical and Computer Engineering Fundamentals,
Rzeszow University of Technology, ul. Wincentego Pola 2, 35-959 Rzeszow, Poland
e-mail: mazur@prz.edu.pl

M. Trojnar
e-mail: trojnar@prz.edu.pl

A. Smoleń
e-mail: smolen@prz.edu.pl

D. Mazur et al. (eds.), *Analysis and Simulation of Electrical and Computer Systems*, Lecture Notes in Electrical Engineering 452,
https://doi.org/10.1007/978-3-319-63949-9_15

The analysis of wind turbines with horizontal axis rotation requires building a 3D calculation model. The building of such a model is very time-consuming and the calculations involving numerical fluid mechanics require substantial calculation power [1].

It is necessary to create geometrical models of turbine rotor elements to be able to conduct CFD analysis. In order to enable horizontal turbine to work properly, it is necessary to choose the right geometrical dimensions of the rotor blade. This requires great knowledge concerning horizontal rotation axis turbine, because from the very stage of designing the blade certain requirements need to be made, such as optimum work at given wind speed or wind conditions. The CFD analysis of the turbine has been conducted with the use of movable net for 3D model [2].

2 CAD 3D Model of Horizontal Rotation Axis Wind Turbine in ANSYS Calculation Environment and Boundary Conditions

The wind turbine blade is one of the most important elements of horizontal rotation axis turbine. The blade geometrical parameters, which have been used to establish the CAD calculation model for the turbine have been presented in Table 1.

Table 1 The blade geometrical parameters for different values of radius

R (m)	Chord profile (m)	Angle of rotation Θ_p (°)
Hub	Hub	Hub
Fixing element—cylinder $\phi = 0.1$	Fixing element	Fixing element
0.29	0.1	0
0.34	0.2	9.9
0.37	0.25	13.4
0.43	0.33	18.07
0.49	0.33	14.29
0.53	0.32	11.91
0.63	0.31	7.98
0.76	0.29	4.72
0.83	0.28	3.42
0.93	0.26	2.08
1.03	0.25	1.12
1.12	0.23	0.49
1.22	0.22	−0.01
1.32	0.21	−0.47
1.42	0.19	−0.92
1.52	0.18	−1.35
1.69	0.16	−1.77

The CAD wind turbine calculation model is presented in Fig. 1.

Numerical modeling of external flow requires establishing finite range/zone [3]. The range/area accepted for calculations must be big enough for established boundary conditions on the boundaries of calculation area not to disturb flow phenomenon near rotor and to present properly the conditions in infinity. A cylinder containing wind turbine has been modeled in order to simulate the rotation of turbine rotor. Figure 2 presents 3D calculation model of the turbine together with external finite area and a cylinder modeling the area near rotor created using ANSYS program [4].

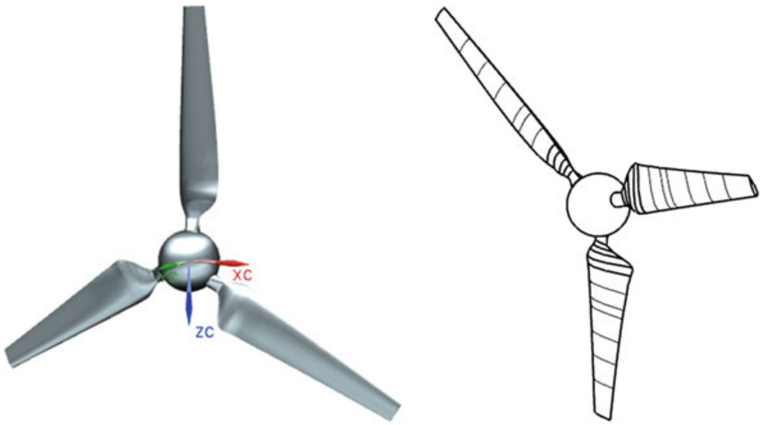

Fig. 1 CAD turbine model of horizontal rotation axis and its view with segment division

Fig. 2 CAD 3D calculation model of turbine with external finite area

3 Discrete Net and Boundary Conditions

Finite area has been modeled in form of a cylinder 6 m in diameter and 20 m long. The movable/mobile AREA/ZONE created in a form of a cylinder 3.6 m in diameter and 0.48 long has a more thickened net compared with the finite area. A set of turbine blades along with turbine hub has been placed inside movable/mobile AREA/ZONE. Moreover, the area near the blades is additionally thickened in order to simulate the behavior of parietal/boundary layer. Figure 3 shows discrete net of calculation model CAD together with the rotor [5].

The boundary conditions assigned to the surfaces limiting the analysis areas have been presented in Fig. 4.

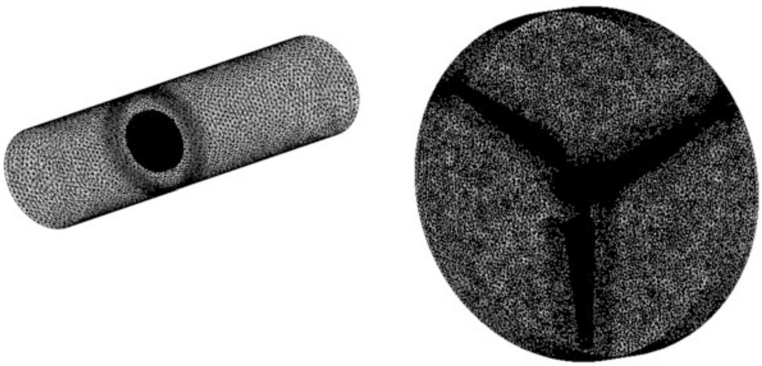

Fig. 3 Discrete net CAD 3D model together with the rotor

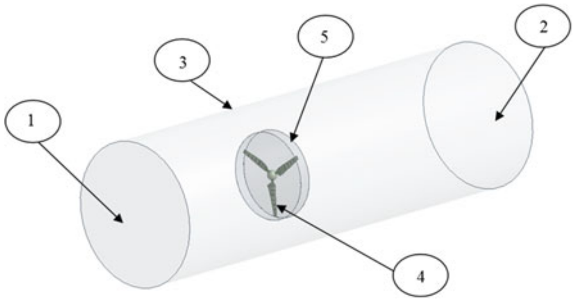

Fig. 4 Boundary conditions assigned to surface; *1* left surface of the cylinder defining condition at the inlet ***of finite area/zone,*** *2* right surface of the cylinder defining condition at the outlet of ***finite area/zone,*** *3* the outer surface of ***finite area/zone*** defined as wall, *4* turbine surface—the conditions of impenetrability, *5* the surface of outer cylinder circle imitating motion and the inner surface of the ***finite area/zone***

Additionally, the following assumptions have been adopted for calculations:

- working pressure 101,325 Pa,
- speed/velocity at inlet 5, 10, 15 m/s,
- viscosity 17.08×10^{-6} kg/ms,
- density 1.225 kg/m^3,
- turbine rotation velocity 200 rpm,
- intensity of turbulence at inlet 1%,
- turbulence mode k- ε - standard.

4 The Results of Calculating Horizontal Rotation Axis Wind Turbine with the Use of Numeric Fluid Mechanics

Figures 5 and 6 present the results of calculations in the form of power lines for the chosen positions of turbine rotor existing in a defined area for flow velocity of 5 and 10 m/s.

Figure 7 presents the calculations results in the form of pressure areas in chosen cutting planes of one of the wind turbine blades at the distance from the middle of coordinate system 0.4; 1; 1.6 m for the inflow velocity of 5, 10 m/s (Fig. 8).

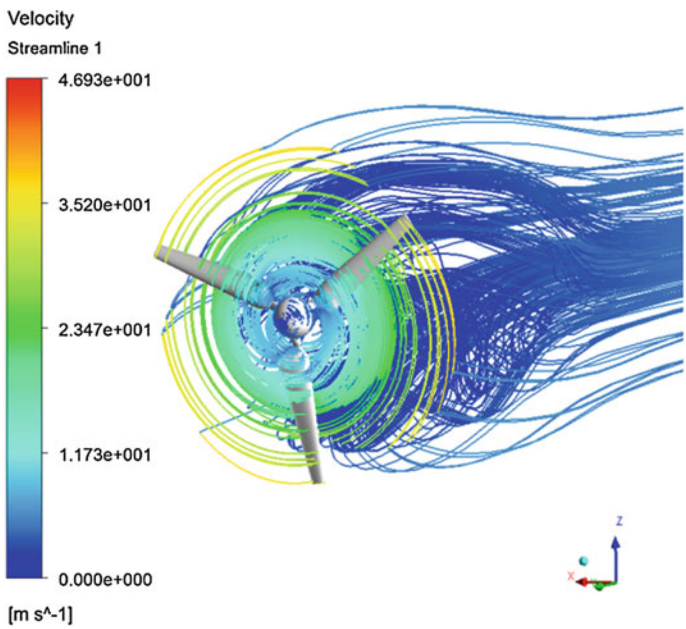

Fig. 5 Power lines in the rotor area for the chosen position of wind turbine rotor at inflow velocity of 5 m/s

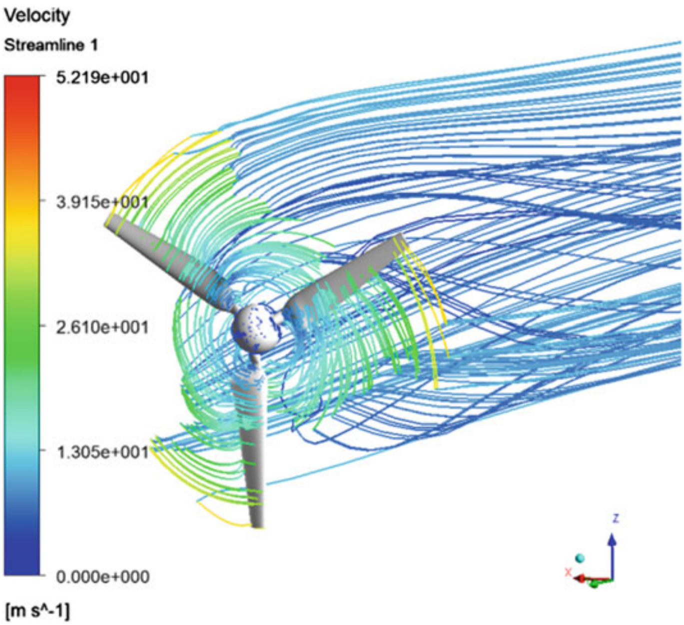

Fig. 6 Power lines in the rotor area for the chosen position of wind turbine rotor at inflow velocity of 10 m/s

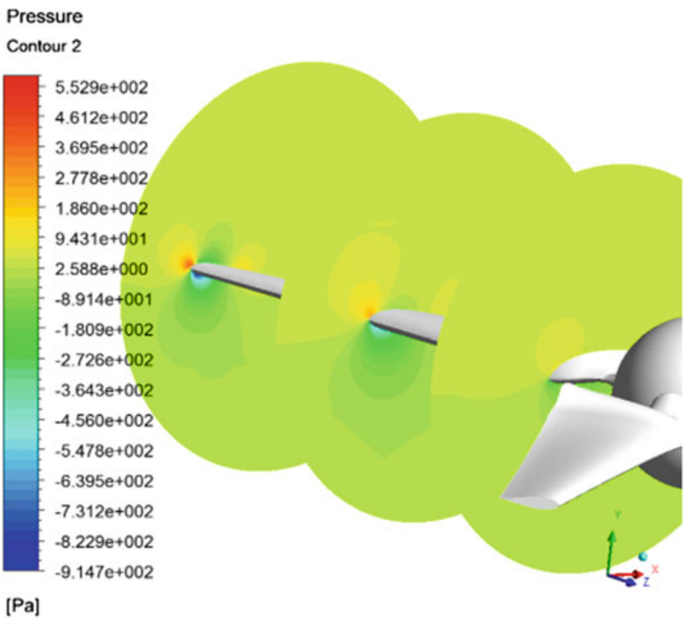

Fig. 7 Pressure areas in cutting planes of a blade for the inflow velocity of 5 m/s

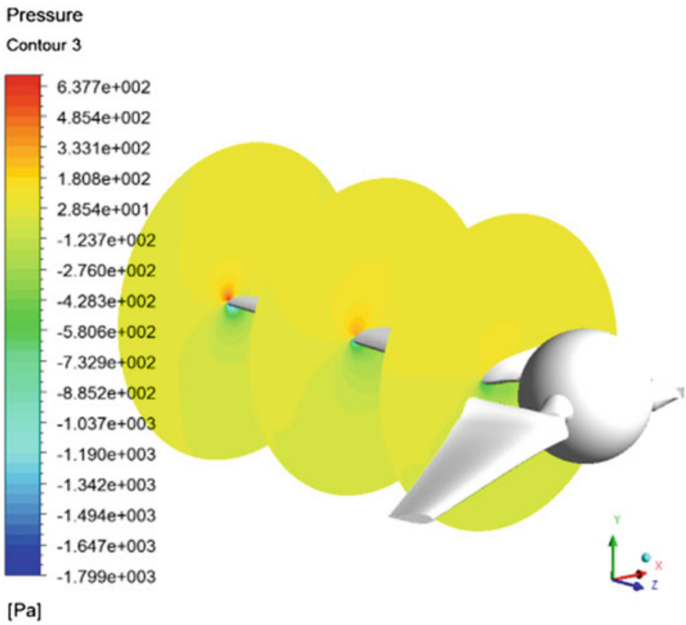

Fig. 8 Pressure areas in cutting planes of a blade for the inflow velocity of 10 m/s

Figures 9 and 10 present the calculations results of the distribution of global velocity on the surface of turbine blades for the chosen positions of turbine rotor and the inflow velocity of 5; 10 m/s.

On the basis of the conducted analysis CFD in Fluent environment, the momentum has been also determined, which consequently allowed the power generated by the wind turbine to be determined. Moreover, analysis has been made for comparative purposes using QBlade environment and BEM method [6].

Figure 11 presents the comparative characteristics of generated power determined using the two methods.

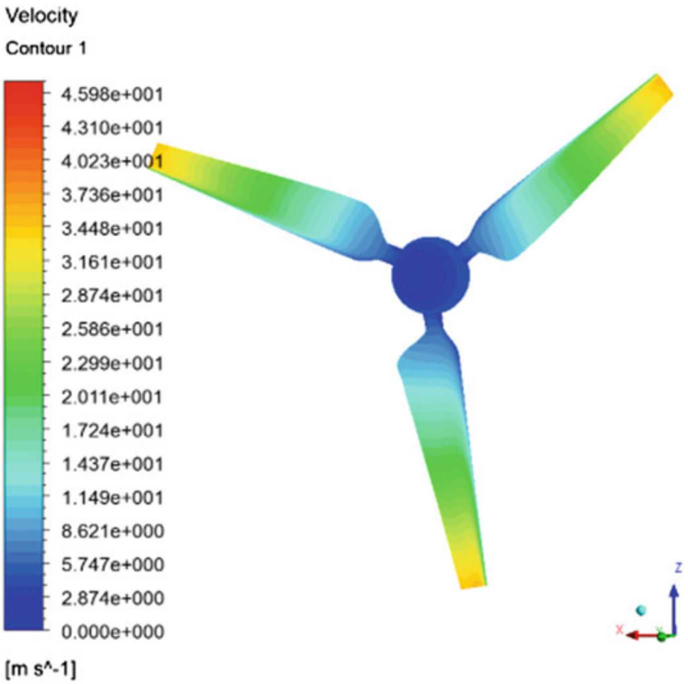

Fig. 9 The distribution of global velocities on the surface of turbine blades for the inflow velocity of 5 m/s

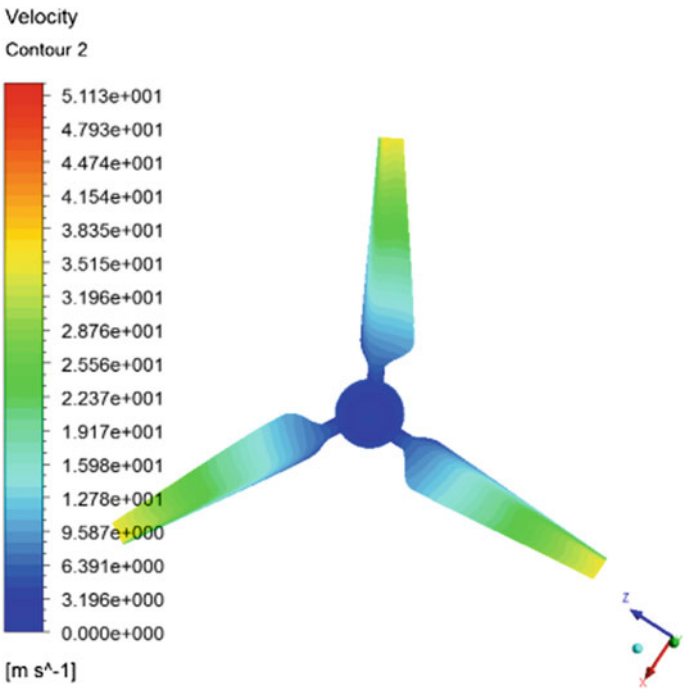

Fig. 10 The distribution of global velocities on the surface of turbine blades for the inflow velocity of 10 m/s

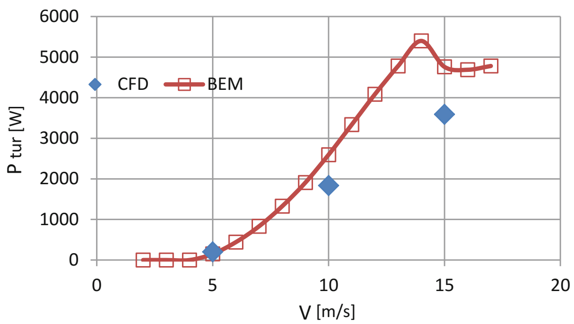

Fig. 11 The comparison of turbine generated power in QBlade program BEM method with that of CFD using numerical fluid mechanics

5 Conclusions

The power values determined by BEM method in QBlade environment for speed over 5 m/s are higher than those determined by CFD method. Exemplary power difference for the velocity of 15 m/s is 32%. CFD method provides complex information concerning the distribution of pressure velocity on the turbine [7]. It is, however, more work consuming and therefore the analysis has been conducted for three velocity values.

References

1. Chi-Jeng, B., Wei-Cheng, W.: Review of computational and experimental approaches to analysis of aerodynamic performance in horizontal-axis wind turbines (HAWTs). Renew. Sustain. Energy Rev. **63**, 506–519 (2016)
2. Fischer, G.R., Kipouros, G.T., Savil, A.M.: Multi-objective optimisation of horizontal axis wind turbine structure and energy production using aerofoil and blade properties as design variables. Renew. Energy **62**, 506–515 (2014)
3. Liu, X., Lu, C., Liang, S., Godbole, A., Chen, Y.: Vibration-induced aerodynamic loads on large horizontal axis wind turbine blades. Appl. Energy **185**(2), 1109–1119 (2017)
4. Nicholls-Lee, R.F., Turnock, S.R., Boyd, S.W.: Application of bend-twist coupled blades for horizontal axis tidal turbines. Renew. Energy **50**, 541–550 (2013)
5. Luo, K., Zhang, S., Gao, Z., Wang, J., Zhan, L., Yuan, R., Fan, J., Cen, K.: Large-eddy simulation and wind-tunnel measurement of aerodynamics and aeroacoustics of a horizontal-axis wind turbine. Renew. Energy **77**, 351–362 (2015)
6. Abdelrahman, M.A., Abdellatif, O.E., Moawed, M., Eliwa, A., Mišák, S.: The CFD performance analysis for horizontal axis wind turbine with different blade shapes and tower effect. In: 16th International Scientific Conference on Electric Power Engineering (EPE) (2015)
7. Abedi, H., Davidson, L., Voutsinas, S.: Numerical studies of the upstream flow field around a horizontal axis wind turbine. In: 33rd Wind Energy Symposium, AIAA SciTech Forum