UDC: 533.682; 533.6.01; 621.548

MODELING AERODYNAMICS OF THE WIND TURBINE WITH ROTATING CYLINDERS

Sakipova S.E.[1], Tanasheva N.K.[2]

[1]E.A. Buketov Karaganda State University, Karaganda, Kazakhstan, sesaule@mail.ru
[2]Institute of Applied Mathematics, Karaganda, Kazakhstan, nazgulya_tans@mail.ru

The paper considers the prospects for the development of alternative energy forms in Kazakhstan, in particular, wind energy. The article discusses the possibilities of using computer simulation of the aerodynamics of the flow around a three-bladed wind turbine by the ANSYS FLUENT software package. The system of initial equations and boundary conditions for solving the problem in axisymmetric formulation are presented. The results of the construction of the computational domain and 3-dimensional grid CAD-model of the wind turbine using the subroutine ANSYS MESH are shown. The analysis of the adequacy of the results of the calculation of the aerodynamic drag force for the virtual model of the wind turbine at various velocities of the incident flow is made.

Keywords*:* wind turbine, aerodynamics, rotating cylinder, drag force, 3-dimensional modeling, Reynolds criterion.

INTRODUCTION

Over recent years, Kazakhstan has been actively developing alternative energy forms, despite the fact that the transition to renewable energy sources for Kazakhstan looks quite difficult due to too cheap coal prices (the country ranks eighth in the world in terms of coal reserves) [1-3]. Indeed, Kazakhstan has significant resources of renewable energy sources (RES), which include water, wind and solar energy. Wind energy is the most relevant direction for the development of renewable energy sources, since wind power plants are among the most environmentally friendly ways to produce energy; they do not require fossil fuels, and do not produce harmful emissions into the environment.

In 2019, the Ministry of Energy of Kazakhstan plans to launch 11 renewable energy source facilities with an overall power of about 400 MW and to generate about 2 billion kWh of electricity [1]. The renewable energy sources market is growing. In 2018, the share of renewable energy in total electricity production was 1.3%, and by 2030 it is planned to reach 10%. Today, intensive work on the design and construction of renewable energy source facilities is in progress region-wise. Favorable conditions are created for the development of renewable energy generating facilities in Kazakhstan, foreign investors are attracted, and auctions are being held to select renewable energy projects [2]. All this necessitates of develop small wind power plants that effectively convert wind energy into electrical or thermal energy.

This paper is concerned with computer simulation of a three-bladed wind turbine with rotating cylinders, which, under certain flow regimes, permits the use of the Magnus effect [4–7]. To determine the most effective parameters, both laboratory and field tests under natural conditions of various wind turbine models were carried out. It is known that the production of mock-ups and the implementation of full-scale and bench tests require significant financial costs and time. At the same time, computing technologies in aerodynamics have reached a high level, which ensures the complementarity of computer simulation and experimental research. The current level of development of software systems, in particular, ANSYS CFX, makes it possible to model a wide class of aerodynamic phenomena from laminar flows to turbulent streams with strong anisotropy of parameters [8, 9]. For example, the transitional model of turbulence correctly analyzes flow regimes that are close to laminar ones, flows with developed flow separation zone and attached flow region.

1. Formulation of the problem.

System of equations

The problem under study at this stage is solved in an axisymmetric formulation, i.e. the dependence on the azimuthal coordinate φ is not taken into account. Fluid flow is assumed to be laminar and is described by a system of control equations in dimensional formulation [10-11], including:

– total energy conservation equation

$$\frac{\partial(\rho h_{tot})}{\partial t}-\frac{\partial p}{\partial t}+\nabla\cdot(\rho U h_{tot})=\nabla\cdot(\lambda\nabla T)+\nabla\cdot(U\cdot\tau)+U\cdot(S_{M,rot}+S_{M,buoy})+S_E \quad (1)$$

– momentum equation

$$\frac{\partial(\rho U)}{\partial t}+\nabla\cdot(\rho U\cdot U)=\nabla p+\nabla\cdot\tau+S_{M,rot}+S_{M,buoy}\ ; \quad (2)$$

– continuity equation

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho U)=0\,; \quad (3)$$

The following symbols are used in the equations (1-3):

$h_{tot}=h_{stat}+U^2/2-\omega^2 r^2/2$ – total enthalpy in rotary mode;

ω —the angular rotation velocity of fluid; r –radius vector; ρ –fluid density;

λ –heat conductivity coefficient; p –pressure; U – velocity vector;

T –temperature; τ –viscous stress tensor; S_E – energy source;

$S_{M,rot}=-2\rho\omega\times U-\rho\omega\times(\omega\times r)$ – the term taking into account the Coriolis force and the centrifugal force;

$S_{M,buoy}=\rho_{ref}\beta(T-T_{ref})g$ – impulse source formed by buoyant forces;

β –volumetric expansion coefficient; g – gravitational acceleration vector;

Boundary conditions:

It is assumed that the adhesion condition holds on the side walls and at the bottom without outlet:

$$U_{wall}=0, \quad (4)$$

At the upper boundary and in the drain region, values of the velocity U and its components along the coordinate axes u_i are given:

$$\begin{cases}U_{inlet}=u_{axial}\bar{i}+u_{radial}\bar{j}+u_{swirl}\bar{k}\\ u_{axial}=-u_1\\ u_{radial}=0\\ u_{swirl}=0\end{cases},\qquad \begin{cases}U_{inlet}=u_{axial}\bar{i}+u_{radial}\bar{j}+u_{swirl}\bar{k}\\ u_{axial}=-u_2\\ u_{radial}=0\\ u_{swirl}=0\end{cases} \quad (5)$$

The computations based on equations (1-3) with boundary conditions (4-5) were performed in the ANSYS CFX and ANSYS FLUENT software package.

.

2. Grid model using the ANSYS MESH subroutine

Previously, an unstructured computational grid of the model was generated in the ANSYS MESH subroutine [9]. As main elements of the computational grid tetrahedrons were chosen; those of the walls were prisms. To simulate the flow in the boundary layer, 18 prismatic layers clustered against the walls were developed. Fig. 1 shows a general view of a two-dimensional computational grid. In the procedure, some general recommendations on generating computational grids used to solve other classes of gas-dynamic problems were taken into account, such as: the two neighboring mesh ratios in the region of greatest gradients should not exceed 1.25, the same at interblock boundaries – 2.0 [8, 9]. After the grid had been generated, its quality was checked. As an estimation parameter, the quality parameter was used, the value of which should be at least 0.1.

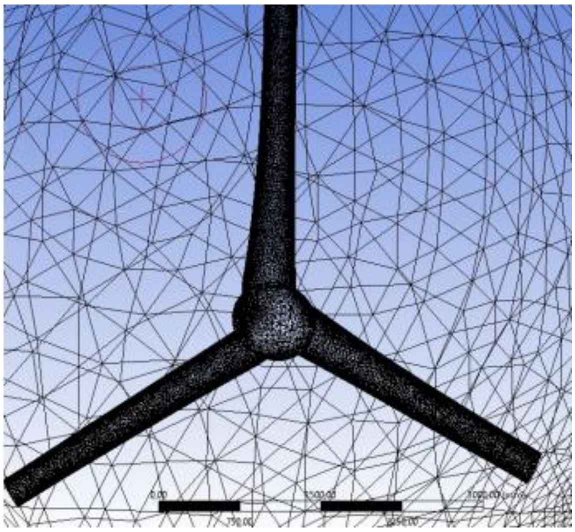

Fig.1. The preliminary 2-dimensional grid model of the wind turbine.

The maximum value of the dimensionless near-wall coordinate Y^{+}_{max} for the grid from the first point was 80.29, and for the grid of the second type it was 102.34. Thus, for the computational domain of the 3-dimensional model, a grid with the number of elements of more than 3,000,000 cells was generated, Fig.2.

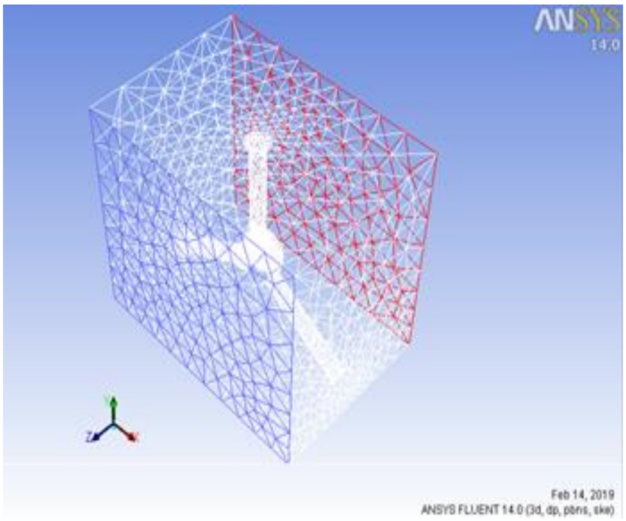

Fig.2. The 3-dimensional grid model of the computational domain around the wind turbine.

Fig. 1 shows that at the initial stage, in the 2-dimensional model the diameter of the cylindrical blades of the wind turbine decreases from their base to the ends, as if they are cut off. As a result of numerous experiments, at the stage of generating 3-dimensional model, the experimenters were able to develop a model of a three-bladed wind turbine with cylindrical blades having flat disks at the ends. Fig.2 most accurately shows the real design [7, see Fig.2]. It can be seen that the diameter of the end bases of the blades is larger than the diameter of the cylindrical blade itself.

Thus, in the ANSYS CFX and ANSYS FLUENT software package, a CAD-model of a wind turbine was developed. That was a wind turbine with blades in the form of rotating cylinders having flat disks at its ends. After that, computational models were developed in DESIGN MODELER. Then, using the mesh generator ANSYS MESH, calculation grids were generated for each element. Next, the computational domains were assembled in the CFX-Pre pre-processor. Figure 3 shows one of the variants of mathematical modeling of a three-bladed wind generator, where the rotation of a cylindrical blade with a flat end is shown.

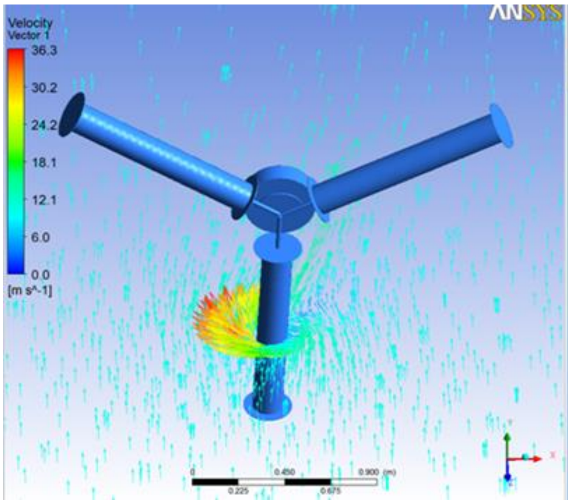

Fig.3. Three-dimensional modeling of the rotation of a wind generator cylindrical blade.

For the numerical implementation of the problem in the described formulation, finite-element software packages ANSYS CFX and ANSYS FLUENT were used. The assumptions about the axial symmetry of the problem and the laminar flow regime permitted to essentially reduce the number of computational nodes and the computation time. In the environment of the same application, the boundary conditions of the problem were set, the accepted assumptions were taken into account and, after completion of the preparation process of objects for modeling, computations could be performed.

3. Discussion of the results

For this particular case, 3.09 million point-elements of the grid model were used in the computations. Computer simulation was made at initial temperature parameters of 288K (15^0C) and 293K (20^0C). The results showed that in this range there was no effect of temperature on the values of the drag force. As a result of the calculations, the authors found the values of the drag force and, accordingly, the drag coefficient when the air flow velocity changed, the maximum value of which was 10m/s. The direction of the incident flow was perpendicular to the plane of the wind wheel and did not change.

Fig. 4 shows the change in the drag force value Fd of the virtual model of the wind turbine at different incident flow velocities. The nature of the dynamics, i.e. an increase in the Fd values corresponds to the physical phenomenon, which is confirmed by experimental data [4-6].

The following known formulas were used to calculate the dimensionless parameter [9]. The formula for determining the Reynolds number:

$$\mathrm{Re} = \frac{ud}{\nu} \tag{6}$$

where u is the velocity of the air flow, approaching the cylinder; d is the outer diameter of the cylinder under study; ν is the kinematic viscosity of air.

The formula for determining the drag coefficient:

$$C_x = \frac{F_d}{\frac{\rho u^2}{2} S} \tag{7}$$

where F_d is the drag force; u is the air flow velocity; ρ is the air density; S – is s the cross-sectional area of the wind wheel.

The found dependence of the drag coefficient of the virtual model of a wind turbine on Reynolds number practically coincides with the exponential law of variation for the ideal case of a cross flow around a cylinder, Fig.5.

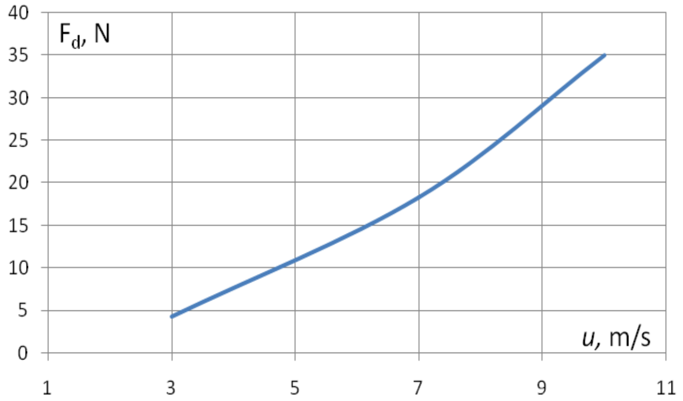

Fig. 4. The dependence of the drag force on the air flow velocity.

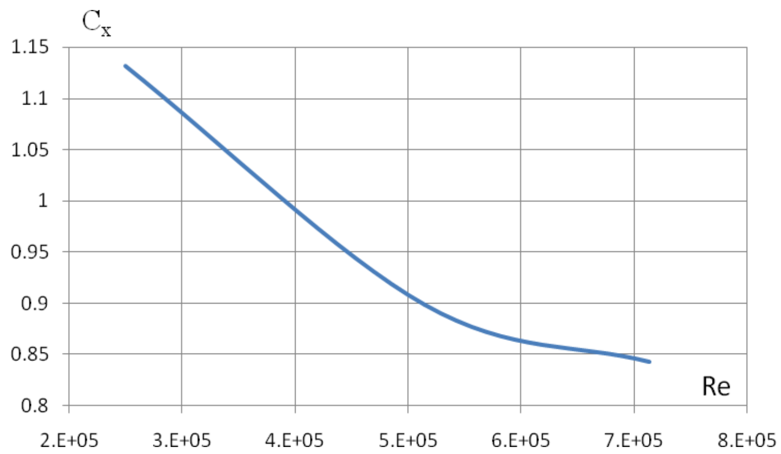

Fig.5. The dependence of the drag coefficient of the wind turbine model on Reynolds number

Thus, the found results of a 3-dimensional modeling of the dynamics of the aerodynamic drag force of a virtual three-bladed model of a wind turbine with rotating cylindrical blades reflect, in a first approximation, the physical flow pattern in the range of air flow variation from 3 to 10 m/s.

Conclusion

At this stage of the simulation, the action of the Magnus effect on the flow aerodynamics has not been studied. Nevertheless, the technique for building a CAD model of a three-bladed wind turbine and simulation of its flow in the ANSYS CFX and ANSYS FLUENT software package has been developed. The ANSYS CFX complex permits to solve complex engineering problems of both internal and external aerodynamics, and to determine the optimal design parameters when exposed to various aerodynamic loads. The calculation of the most efficient flow parameters of the wind turbine will be carried out in ANSYS CFX with corrected graphical parameters of the model in a wider range of boundary conditions.

Acknowledgements

This research was funding by Ministry of Education and Science of the Republic of Kazakhstan of the Program AP051315203 "Development and prototyping wind power plant for alternative power supply using a domestically produced electric generator" for 2018-2020.

REFERENCES

1 Ignatieva A. IA Neftegaz.RU, Astana, 11/02/2019. Available at: https://neftegaz.ru /news/ Alternative-energy/194767

2 The development of green energy in Kazakhstan. Available at: www.eenergy.media/2018/09/19/ razvitie-zelyonoj-energetiki-v-kazahstane/

3 Karatayev M., Clarke M. A review of current energy systems and green energy potential in Kazakhstan. *Renewable and Sustainable Energy Reviews*, Vol. 55, March 2016, pp. 491–504.

4 Kussaiynov K., Sakipova S.E., Tanasheva N.K., Kambarova Zh.T., et al. Wind turbine based on the Magnus effect. Innovative patent №30462 of 23.09.2015.

5 Sakipova S.E., Tanasheva N.K., Kivrin V.I., Kussaiynova A.K. Study of wind turbine model aerodynamic characteristics with rotating cylinders». *Eurasian Physical Technical Journal*. 2016, Vol.13, No.2 (26), pp.112 – 117.

6 Tanasheva N.K., Shrager E.R., Sakipova S.E. et al. Research of aerodynamic characteristics of the wind generator on the basis of Magnus's effect. *Bulletin of the Karaganda University*. *Physics Series*. 2017, No. 3(87), pp. 60 – 64.

7 Sakipova S.E., Shrager E.R., Tanasheva N.K., Botpayev N., Kussaiynova A.K. Study of electro-physical characteristics of wind turbines. *Eurasian Physical Technical Journal*. 2018, Vol.15, No. 2(30), pp.117 – 122.

8 ANSYS Fluent software. Available at: https://www.ansys.com/products/fluids/ansys-fluent

9 Computer modeling of 3D-models of aviation equipment and engineering calculations. Available at: http://www.ipmce.ru/custom/vsop/themes/3dmodel/

10 Martin O. L. Hansen L. *Aerodynamics of Wind Turbines*. London, Sterling, VA, 2008, 208 p.

11 Krasnov N.F. Aerodynamics. Part 2. *Aerodynamical methodology*. Moscow, Higher school, 1980, 416 p.

Article accepted for publication 29.05. 2019

.