DOI 10.31489/2022PH3/108-114

UDC 533.682

A.R. Bakhtybekova[1,2*], N.K. Tanasheva[1,2], N.N. Shuyushbayeva[3], L.L. Minkov [4], N.K. Botpaev[1]

[1] Karaganda University of the name of academician E.A. Buketov, Karaganda, Kazakhstan
[2]Scientific Center "Alternative Energy", Karaganda, Kazakhstan
[3] Sh.Ualikhanov Kokshetau University, Kokshetau, Kazakhstan
[4] National Research Tomsk State University, Tomsk, Russia
(**E-mail:* asem.alibekova@inbox.ru)

Analysis of velocity and pressure vector distribution fields in a three-dimensional plane around a wind power plant

To date, there has been an increase in demand for electric energy obtained from clean renewable energy sources. One of them is wind power. Based on this, the development and research of new types of efficient wind turbines that start working at low wind speeds is an urgent issue. Wind turbines operating based on the Magnus effect have proven their effectiveness. However, the authors of this work, for the first time, to eliminate the problem in the form of an electric drive for the promotion of cylindrical blades added a deflector element to the end of the cylinders. Before creating an experimental setup, it is necessary to numerically investigate the aerodynamics around the wind wheel. For this purpose, numerical simulation of wind wheel aerodynamics has been carried out using the highly efficient Ansys Fluent program. A three-dimensional geometry has been created in Design Modeler. A mathematical model grid with a grid number of 47329 consisting of tetragonal cells is constructed. The Realizable k-ε is chosen as the turbulence model. A thorough analysis of the velocity vector distribution fields for flow and pressure velocities in the three-dimensional plane around the wind wheel at air flow velocities of 5.10 and 15 m/s is carried out.

Keywords: wind power plant, Ansys Fluent, Magnus effect, deflector, mathematical model, numerical simulation.

Introduction

Currently, many problems faced by engineers and researchers in the field of aerodynamics are not amenable to experimental solutions or require huge material and human resources. An alternative solution in this case is the use of numerical research based on computer programs.

Computational fluid dynamics (CFD) is a branch of science that solves the problem of modeling heat and mass transfer in various technical and natural objects. The main task of CFD is the numerical solution to the Navier Stokes equations describing fluid dynamics. These equations constitute a mathematical model of heat and mass transfer [1, 2].

For aerodynamic practice, an important role is played by the data of air flow conditions of cylindrical bodies of various configurations.

The number of papers devoted to CFD in the field of wind energy is growing every year. In [3], the authors performed a high-resolution simulation of the flow around a wind power plant and its blades.

One of the representatives of wind turbines of installations showing their efficiency starting from 2-4 m/s is wind turbines operating based on the Magnus effect. Along with the obvious advantages over traditional blade wind turbines, to increase the performance indicators, it is necessary to improve the shape and parameters of the working power elements - the blades of the installation. The authors of [4] conducted a 3D numerical study of a Magnus-type wind turbine equipped with cylindrical blades with different aspect ratios. The analysis of the influence of various shapes and lengths of blades on the performance of the Magnus wind turbine is carried out.

However, existing wind turbines operating based on the Magnus effect have a disadvantage in the form of an additional source of electric drive for the promotion of cylindrical blades [5]. Accrodingly, we add a deflector to exclude the electric drive, as well as the disruption of the air flow from the ends of the cylinders.

The purpose of this work is to create a mathematical model of a wind turbine with two blades in the form of rotating cylinders with a deflector.

Mathematical model

The simulation was carried out using the Ansys Fluent software package. Structurally, the numerical simulation is shown in Fig. 1.

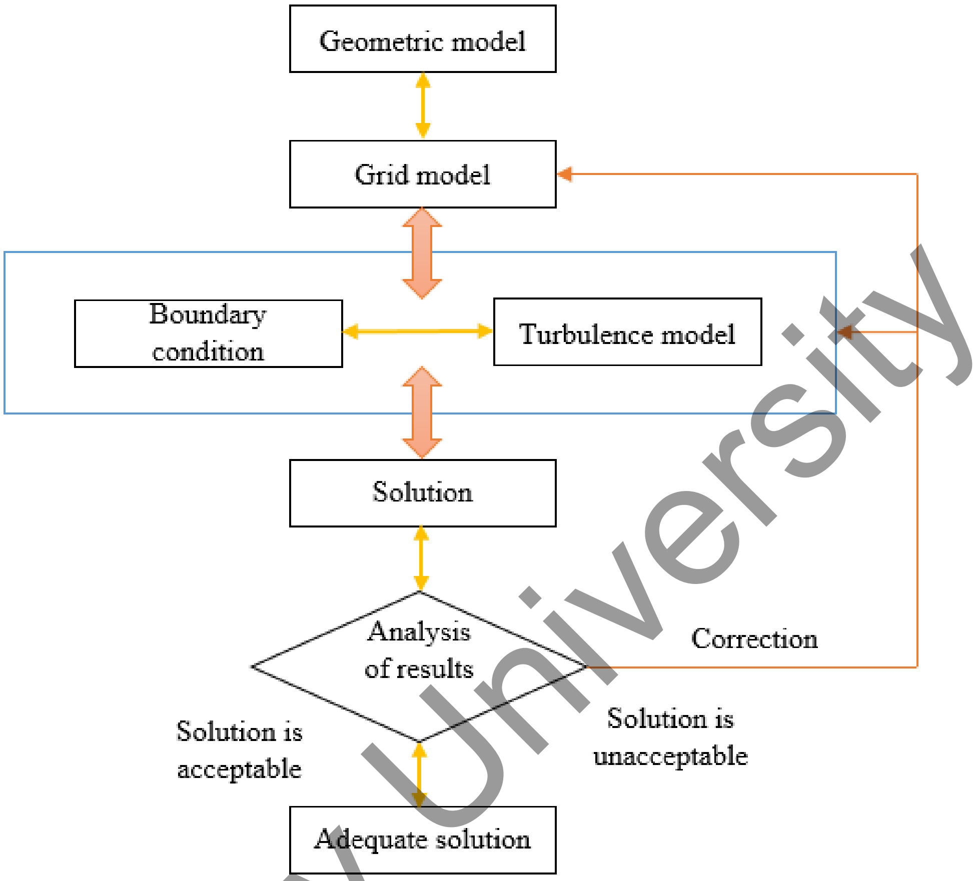


Figure 1. Structure of execution, numerical simulation

The first stage in the numerical simulation process of a wind power plant is the construction of the geometry of a wind power plant using the COMPASS 3D program. A three-dimensional solid-state model of a wind power plant with 2 blades in the form of rotating cylinders with a deflector created using the COMPASS 3D program is shown in Figure 2.

In this case, we consider the flow of air around a wind wheel consisting of two blades in the form of a rotating cylinder with a deflector located in the XY plane at an angle of 180 ° to each other relative to the z-axis of rotation of the wind wheel (Figure 3). The axes of rotation of the cylinders are in the XY plane. The axis of rotation of the wind wheel coincides with the Z axis.

The entire working area was divided into three types of nested subdomains (Fig. 3): subdomains of the 1st type (cylinders), built around the working blades of the wind wheel and rotating at the speed of the working cylinders (1); subdomain of the 2nd type (sphere), built around the wind wheel minus cylindrical subdomains 1-type (2); type 3 subdomain (cube) surrounding type 2 subdomain minus (3).

The radius of the outer cubic subdomain (3) is assumed to be 0.2 m, spherical subdomain (2) has a radius of 0.1 m, cylindrical subdomains (1) have a radius of 0.02 M.

The boundary conditions in the form of the incoming velocity of the incoming air flow are set on the front wall, and the outlet pressure is set on the back wall of the cubic subdomain. The remaining walls are set to the walls of symmetry (Figure 3).

The next step in the process of numerical modeling is to create a computational grid for this task in Ansys Meshing with a minimum set of actions.

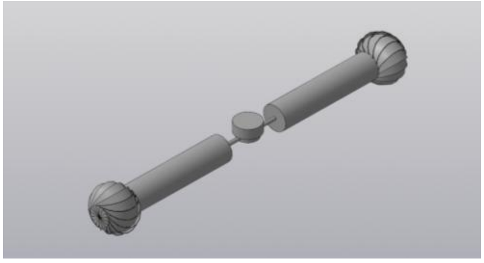


Figure 2. Three-dimensional geometry of a wind power plant with 2 blades created in the COMPASS 3D program

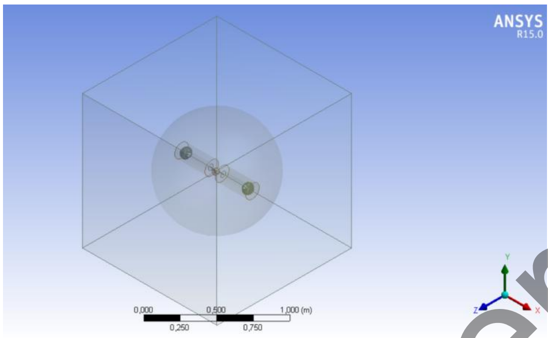


Figure 3. Calculation area

A finite-volume grid constructed in subdomains of type 1, 2, 3 consists of tetragonal cells. Figure 4 represents the grid view in the z=0 plane. The grid is depicted in the XY plane, the cross section of the Z=0 area. The total number of cells is 47329.

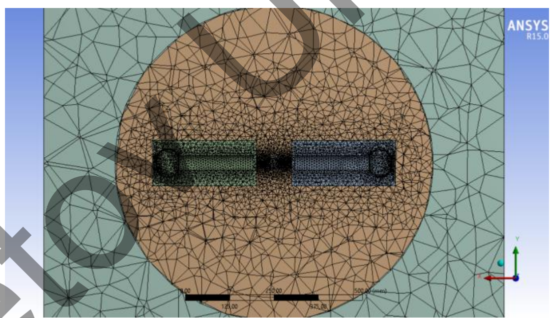


Figure 4. Finite-volume grid

The Realizable k-ε model was chosen as the turbulence model, which gives a general description of turbulence using two transport equations. This model is a widely used model in the field of computational fluid dynamics (CFD) and is used to model average flow characteristics for turbulent flow conditions.

Table 1 presents the boundary conditions specified in the numerical study.

Table 1

List of boundary conditions

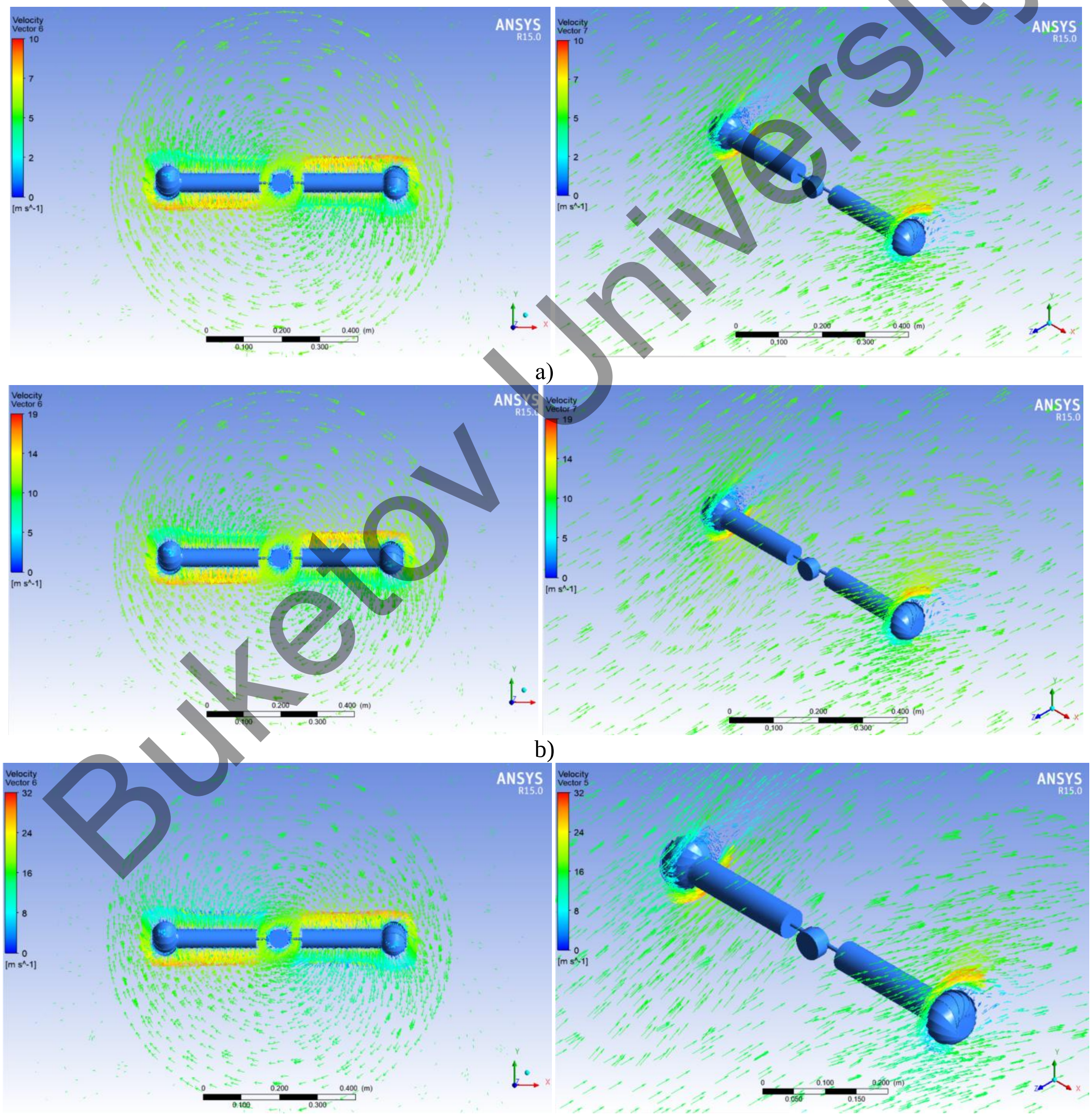
Boundary conditions	
At the entrance	
1	2
View	Speed at the entrance
Initial manometric pressure (Pa)	0
Air flow velocity, m/s	3,5,7,10,12,15
Turbulence intensity (%)	5
Coefficient of turbulent viscosity	10
At the exit	
View	Outlet pressure

1	2
Manometric pressure (Pa)	0
Return flow of turbulent intensity (%)	5
Return flow coefficient of turbulent intensity (%)	10
Blade surface	
View	Wall
Shift Condition	No sliding

Results of mathematical modeling of a wind power plant with 2 blades

To calculate aerodynamic characteristics and mathematical modeling, a wind power plant with 2 blades in the form of rotating cylinders with a deflector, created on the basis of the Magnus effect, is considered.

The COUPLED scheme was used to coordinate the pressure field and the velocity field. Time derivatives were resolved with the second order of accuracy. Figure 5 presents the results obtained by numerical investigation of the distribution field of the velocity vectors of the incoming air flow around the wind wheel of a wind turbine with 2 blades at speeds of 5.10 and 15 m/s.



a)

b)

c)

Figure 5. Velocity vector distribution fields in a three-dimensional plane around a wind wheel:
a) at v = 5 m/s; b) at v = 10 m/s; c) at v= 15 m/s.

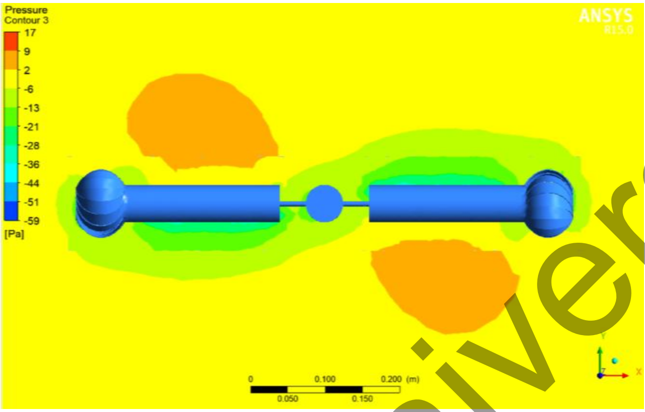
As can be seen from Figure 5, there is a complex three-dimensional nature of the wind wheel flow. The direction of rotation of the cylinder is set on the x-axis and the wind wheel on the z-axis.

The deformation of the velocity vector distribution field in the three-dimensional plane occurs because of an increase in the air flow velocity on one side of the blade and a decrease in the flow velocity on the other side caused by the rotation of cylinders with deflectors around their axes in the positive direction.

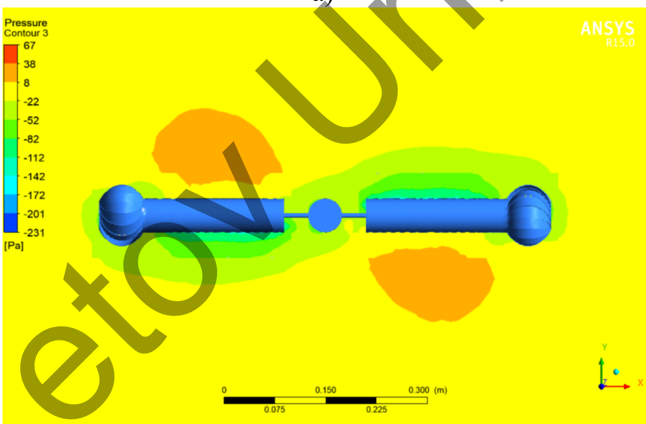
It can be seen that due to the unfavorable pressure gradient, the boundary layer around the surface of the blades separates with an increase in the velocity of the incoming air flow.

It was also found that by adding a deflector to the end part of the cylinder, the aerodynamics around the cylinder improved, the disruption of the airflow from the ends of the cylinders was eliminated.

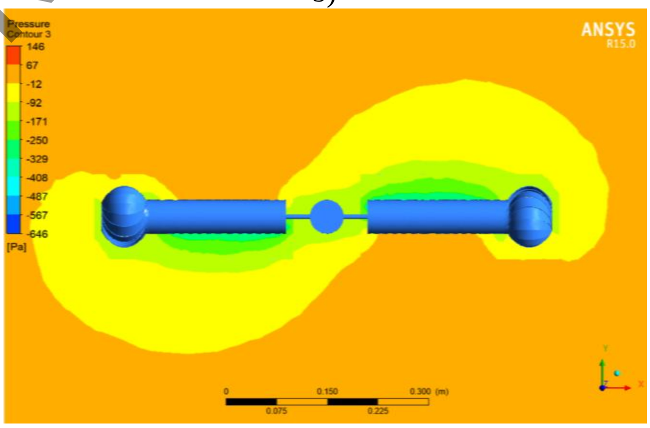
Figure 6 illustrates the static pressure distributions (pct = p – ratm) in the vicinity of the wind wheel (plane z= 0) obtained for different speeds of the incoming flow (wind).



a)



b)



c)

Figure 6. Pressure distribution fields in the three-dimensional plane around the wind wheel:
a) at v = 5 m/s; b) at v = 10 m/s; c) at v= 15 m/s

The rotation of the blades in the conditions of an incoming flow leads to the fact that on one side of the cylinder with a deflector, the air velocity will be greater than on the other side. According to Bernoulli's law, in the area where the flow velocity is higher, the pressure becomes lower. Therefore, on one side of the blade (in the figures shown left side), the pressure is lower than on the other, resulting in a force (lifting) acting on each blade, which is directed perpendicular to the axis of the blade and the direction of the wind. Since all cylinders rotate in the same direction (clockwise) relative to their own axes, the lifting forces will create a moment of forces that causes the wind wheel to rotate clockwise relative to the z axis. The results obtained do not contradict the results of the authors [6, 7].

At an incoming flow velocity of 5 m/s, the static pressure in the vicinity of the rotating wind wheel varies from -59 Pa to 17 Pa. An increase in wind speed leads to an expansion of the range of pressure changes. So for a wind speed of 10 m/s, this range is from – 231 Pa to 67 Pa, for 15 m/s – from -646 Pa to 146 Pa.

Conclusions

The authors found that by adding a deflector to the end zone of the cylinder, the aerodynamics around the cylinder improved, the disruption of the air flow from the ends of the cylinders was eliminated.

In the course of performing a numerical study of aerodynamics around a rotating wind wheel of a wind power plant with 2 blades in the form of rotating cylinders with a deflector:

– a three-dimensional geometry of a wind power plant with 2 blades created in the COMPASS 3D program was created;

– a mathematical model grid with a grid number of 47329 consisting of tetragonal cells was constructed;

– selected as a Realizable k-ε turbulence model, which improved characteristics compared to the standard k-ε model when applied to flows involving boundary layers with strong unfavorable pressure gradients;

– velocity vector distribution fields were obtained for flow velocities of 5.10 and 15 m/s, during which it was determined that due to an unfavorable pressure gradient, the boundary layer around the surface of the blades separates with an increase in the velocity of the incoming air flow;

– pressure distribution fields were obtained in a three-dimensional plane around the wind wheel for flow velocities of 5.10 and 15 m/s, at which it was determined that the rotation of the blades in the conditions of an incoming flow leads to the fact that the air velocity on one side of the cylinder with a deflector will be greater than on the other side.

References

1 Tanasheva, N.K., Bakhtybekova, A.R., Shaimerdenova, G.S., Sakipova, S.E., Shuyushbaeva, N.N. (2022). Modeling Aerodynamic Characteristics of a Wind Energy Installation with Rotating Cylinder Blades on the Basis of the Ansys Suite. *Journal of Engineering Physics and Thermophysics*. https://doi.org/10.1007/s10891-022-02500-3

2 Tanasheva, N.K., Bakhtybekova, A.R., Sakipova, S.E., Minkov, L.L., Shuyushbaeva, N.N., & Kasimov, A.R. (2021). Numerical simulation of the flow around a wind wheel with rotating cylindrical blades. *Eurasian Physical Technical Journal*, 18, 1(35), 51–56. https://up.ksu.kz/phtj/2021_18_1_35/7.pdf

3 Miller, A., Chang, B., Issa, R., & Chen, G. (2013). Review of computer-aided numerical simulation in wind energy. *Renewable and Sustainable Energy Reviews*, 25, 122–134.

4 Sun, X., Zhuang, Y., Cao, Y., & Huang, D. (2012). A three-dimensional numerical study of the Magnus wind turbine with different blade shapes. *Journal of Renewable and Sustainable Energy*, 4, 063139. https://doi.org/10.1063/1.4771885

5 Tanasheva, N.K., Bakhtybekova, A.R., Minkov, L.L, Bolegenova, S.A., Shuyushbaeva, N.N., Tleubergenova, A.Zh., & Toktarbaev, B.A. (2021). Influence of a rough surface on the aerodynamic characteristics of a rotating cylinder. *Bulletin of the university of Karaganda-Physics*, 3(103), 52–59.

6 Wang, M., Avital, E.J., Chunning, Ji X.B., Xu, D., Williams, J.J.R., & Munjiza, A. (2020). Fluid–structure interaction of flexible submerged vegetation stems and kinetic turbine blades. *Computational Particle Mechanics*, 7, 839–848. https://doi.org/10.1007/s40571-019-00304-6

7 Kui, O., Chunlei, L., Sachin, P., & Antony, J. (2009). High-Order Spectral Difference Simulation of Laminar Compressible Flow Over Two Counter-Rotating Cylinders . *27th AIAA Applied Aerodynamics Conference* (22-25 June 2009). San Antonio (Texas). https://doi.org/10.2514/6.2009-3956

А.Р. Бахтыбекова, Н.К. Танашева, Н.Н. Шуюшбаева, Л.Л. Миньков, Н.К. Ботпаев

Жел энергетикалық қондырғысының айналасындағы үш өлшемді жазықтықтағы жылдамдық пен қысым векторларының таралу өрістерін талдау

Бүгінгі таңда таза жаңартылатын энергия көздерінен алынған электр энергиясына сұраныстың өсуі байқалады. Солардың бірі — жел энергетикасы. Осыған сүйене отырып, желдің төмен жылдамдығымен жұмыс істей бастайтын тиімді жел энергетикалық қондырғысының (ЖЭҚ) жаңа түрлерін әзірлеу және зерттеу өзекті мәселе болып табылады. Магнус эффектісі негізінде жұмыс істейтін ЖЭҚ тиімді екенін дәлелдеді, бірақ бұл жұмыстың авторлары электр жетегі түрінде мәселені шешу үшін алғаш рет цилиндр қалақшаларын айналдыру үшін цилиндрлердің соңына дефлектор элементін қосты. Эксперименттік қондырғыны жасамас бұрын жел доңғалағының айналасындағы аэродинамиканы сандық түрде зерттеу қажет. Осы мақсатта Ansys Fluent жоғары тиімді бағдарламасын пайдалана отырып, жел доңғалағының аэродинамикасына сандық модельдеу жүргізілді. Design Modeler-де үш өлшемді геометрия жасалды. Тетрагональды ұяшықтардан тұратын тор саны 47329 болатын математикалық модельдің торы құрастырылды. Турбуленттілік моделі ретінде Realizable k-ε таңдалды. Ауа ағынының жылдамдығы 5, 10 және 15 м/с болатын жел доңғалағының айналасындағы үш өлшемді жазықтықтағы ағын жылдамдығы мен қысымы үшін жылдамдық векторларының таралу өрістеріне мұқият талдау жүргізілді.

Кілт сөздер: жел энергетикалық қондырғы, Ansys Fluent, Магнус эффектісі, дефлектор, математикалық модель, сандық модельдеу.

А.Р. Бахтыбекова, Н.К. Танашева, Н.Н. Шуюшбаева, Л.Л. Миньков, Н.К. Ботпаев

Анализ полей распределения векторов скоростей и давления в трехмерной плоскости вокруг ветроэнергетической установки

К сегодняшнему времени наблюдается рост спроса в электрической энергии, полученной из чистых возобновляемых источников энергии. Одним из них является ветроэнергетика. Исходя из этого, разработка и исследования новых видов эффективных ветроэнергетических установок (ВЭУ), которые начинают работать при малых скоростях ветра, является актуальным вопросом. ВЭУ, работающие на основе эффекта Магнуса, доказали свою эффективность, однако авторами данной работы впервые для устранения проблемы в виде электрического привода для раскрутки цилиндрических лопастей на конец цилиндров добавлен элемент — дефлектор. Перед созданием экспериментальной установки необходимо численным путем исследовать аэродинамику вокруг ветроколеса. С этой целью проведено численное моделирование аэродинамики ветроколеса, используя высокоэффективную программу Ansys Fluent. Создана трехмерная геометрия в Design Modeler. Построена сетка математической модели с числом сетки 47329, состоящей из тетрагональных ячеек. В качестве модели турбулентности выбрана Realizable k-ε. Проведен тщательный анализ полей распределения векторов скоростей для скоростей потока и давления в трехмерной плоскости вокруг ветроколеса при скоростях воздушного набегающего потока 5, 10 и 15 м/с.

Ключевые слова: ветроэнергетическая установка, Ansys Fluent, эффект Магнуса, дефлектор, математическая модель, численное моделирование.