UDC 533.6.01; 621.548

N.K. Tanasheva[1], E.R. Shrager[2], A.N. Dyusembaeva[1],
A.K. Kussaiynova[1], D.A. Ospanova[1], A.R. Bakhtybekova[3]

[1]Ye.A. Buketov Karaganda State University;
[2]Tomsk State University, Russia;
[3]M.O. Auezov South Kazakhstan State University, Shymkent
(E-mail: nazgulya_tans@mail.ru)

Simulation of flow of the blade wind turbine

The paper is concerned with simulation of aerodynamics of a wind turbine blade in 3D space. The authors developed a grid model of the airspace in the form of an infinite cylinder wherein a wind turbine blade is arranged. The simulation made it possible to draw a picture of the cross-flow of a sail blade at various airflow rates. The investigators determined the pressure distribution pattern and distribution of the current lines in the symmetric plane for the angle of attack $\alpha=0°$ at an approach flow rate of 5 m/s.

Key words: Ansys Fluent, blade, flow pattern, simulation.

Introduction

The investigation process of simulation of the flow of a triangular sail blade by an airflow using Fluent software package includes the following solution stages: the development of the computational model, the finite-difference grid plotting, running the Ansys Fluent solver, processing of the results [1].

The computational model development stage is one of the most important steps to achieve a successful solution of the problem. Designed in the right way computational model, reasonably broken down into finite elements, significantly increases the quality of the solution of the problem, reducing the probability of occurrence of nonphysical picture of the flow and contributes to the achievement of results close to experimentally obtained ones. The construction of the model geometry can be done in two ways:

1) Development of the geometry using the internal means of the program;
2) Development of the model in CAD programs and their further importing.

To build a two-dimensional model of the wind turbine blade an internal program of the Ansys Fluent, i.e. Gambit was used [2–4].

Problem statement

To develop a three-dimensional model of the wind turbine blade an AutoCAD (Autodesk) software package was applied (Fig. 1). The blade was a right isosceles triangle, which was captured when it was ultimately blown by the airflow. The area surrounding the sail corresponded to the dimensions of the wind tunnel T-1-M [5].

To proceed to the next stage, the investigators exported the resulting geometric model to the Gambit program and they built a finite-difference grid based on the model. They also checked the quality of the grid and set the boundaries of the computational domain, within which the boundary conditions would be established later. Then the export of the finite-difference grid to the Fluent program was carried out.

To determine the effect of a difference grid size (number of cells) of the two-dimensional model of the blade on the drag force, calculations for three difference grids were made using the model. Table 1 shows the numerical comparison of drag force results obtained for various difference grids. The value of F_1 corresponds to the drag force obtained using the model on the grid of 20.000 nodes, F_2 — of 40.000 nodes, and F_3 — of 160.000 nodes.

a) a wireframe view *b*) a volumetric view

Figure 1. The three-dimensional model of a wind turbine blade

Table 1

Comparison of drag forces for various difference grids

Velocity, m/s	Drag force		
	F_1, N	F_2, N	F_3, N
4	1.49	1.56	1.56
5	1.76	2.40	2.42
6	3.10	3.25	3.26

Table 1 shows that value differences of drag forces for grids of 40000 and 160000 nodes are almost minimal, and the difference grid of 20000 nodes is much different.

Therefore, for a two-dimensional model of the triangular sail blade of the wind turbine it will be more reasonable to use a grid of 40000 nodes later on (Fig. 2).

To determine the effect of the size of the difference grid of the three-dimensional model of the blade on the drag force, using the model the authors made calculations for two difference grids: 1 — for 1000000 nodes, 2 — for 1400000 nodes.

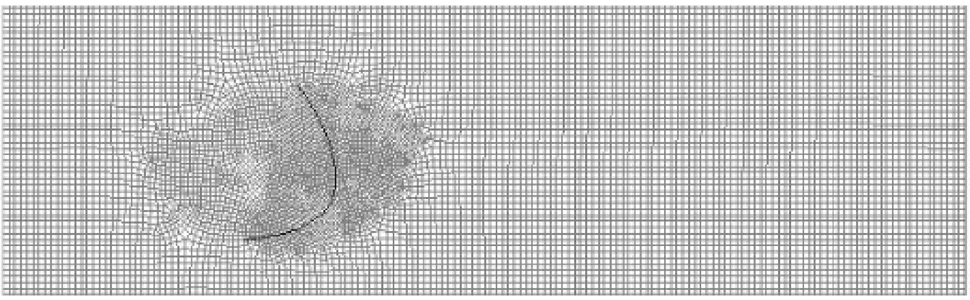

Figure 2. Finite-difference grid of the sail blade of the wind turbine

Table 2 shows the results of calculations of drag forces for various finite-difference grids. The value of F_1 corresponds to the drag force obtained using the model on the grid of 1000000 nodes, F_2 — of 1400000 nodes, and F_3 — is obtained experimentally.

Table 2

Comparison of drag forces for various difference grids

Velocity, m/s	Drag force		
	F_1, N	F_2, N	F_3, N
4	1.49	1.56	1.55
6	3.46	3.26	3.28
8	5.10	5.76	5.78

Table 2 shows that the values of the drag force obtained by means of experiments and numerical simulation method, on the finite-difference grid of 1,400,000 cells are slightly different. In this regard, for further calculations the experimenters will use the grid of 1,400,000 cells (Fig. 3).

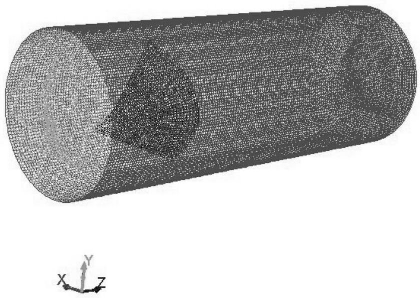

Figure 3. The finite-difference grid of a three-dimensional model of a wind turbine blade

The stimulation by Ansys Fluent is based on the solution of the Navier-Stokes equations of energy and continuity. The system of equations describing the flow of gas in vector form is represented as follows.

The numerical simulation was carried out based on solving two-dimensional equations [6] with boundary conditions (1)–(3) using Patankar method, implicit scheme of the second order space accuracy for convective terms of equations, two-parameter model of turbulence k-ε.

The boundary conditions at the *input* bound:

$$U = U_{in}\,;\ V = 0\,. \tag{1}$$

The boundary conditions at the *output* bound:

$$\frac{\partial \varphi}{\partial x} = 0\,. \tag{2}$$

For the k-ε-model the investigators used a standard recommended set of empirical constants (2.0), which is usually defaulted in computational packages:

$$C_{\mu} = 0.09,\quad C_{\varepsilon 1} = 1.44,\quad C_{\varepsilon 2} = 1.92,\quad \sigma_k = 1.0,\quad \sigma_{\varepsilon} = 1.3\,. \tag{3}$$

The dimensions of the computational domain were set in accordance with the measures of the wind tunnel.

Results and discussion

The next stage of the solution was processing of the calculation results. The authors determined the fields of rates, of the pressure distribution and of the current lines in the symmetric plane for the angle of attack α=0° at an approach flow rate of 5 m/s (Fig. 4).

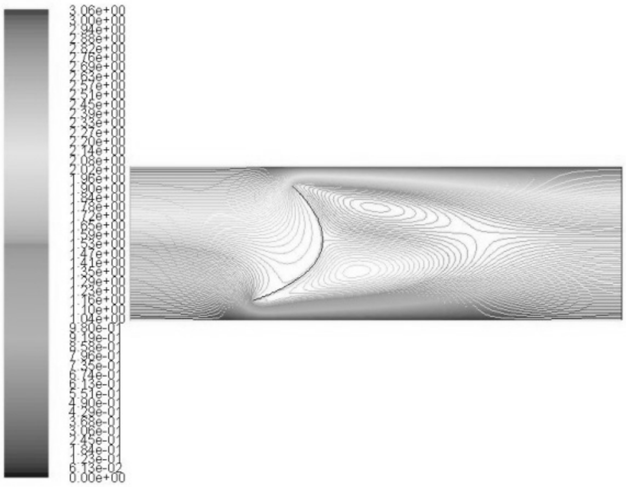

Figure 4. The field of the current lines distribution

Figure 4 shows that the blade tips are high pressure zones and the depression zones are located outside of the sail. It is seen that behind the sail owing to the formation of circulating zones a backflow occurs, which may be caused by the capture of the air flow by the boundary layer.

Conclusion

Thus, the authors made the analysis of the development of the computational model of a sail blade in an airflow. Using ANSYS FLUENT software, the authors determined the pattern of the cross flow of the sail blade at a variety of cross flow rates.

On the basis of numerical simulation, universal dependences of the aerodynamic parameters on the geometry of the blade profile have been established for various velocities of the wind flow. The results of simulation of the flow past a triangular sail-type blade have been obtained. The regularities in the variation of the aerodynamic parameters established in this study can be useful for understanding the complex aerodynamic pattern of the turbulent air flow past bodies with various profiles. The universal dependences obtained for the driving force and drag can be used in designing sail-type wind turbines.

References

1 *Valiev M.J.* Calculations low-speed wind turbines with high energy utilization factor of wind // XX Tupolev's readings: Proceedings of the International Youth Scientific Conference. — Kazan, 2013. — P. 23–26.

2 *Isaev S.A., Baranov P.A., Mitrofovich A.E., Kolosov V.V., Ponomarev A.D.* Numerical simulation of turbulent flow within the wind turbine based power impeller // Journal of Engineering Physics. — 2003. — Vol. 76, No. 6. — P. 45–48.

3 *Baturin O.V., Baturin N.V., Matveev V.N.* Construction of settlement models in the preprocessor Gambit Fluent universal software system: Proc. Benefit. — Samara, 2009. — 172 p.

4 *Baturin O.V.* Calculation of liquid and gas flows using the Fluent Universal software: Textbook. — Samara, 2010. — 151 p.

5 *Kussaiynov K., Sakipova S.E., Kambarova Zh.T., Turgunov M.M.* The development of a double wind turbine with sail wind wheels placed at right angle to one another // Eurasian Physical Technical Journal. — 2015. — Vol. 12, No. 1(23). — P. 53–58.

6 *Kussaiynov K., Tanasheva N.K., Min'kov L.L., Nusupbekov B.R., Stepanova Yu.O., Rozhkova A.V.* Numerical simulation of a flow past a triangular sail-type blade of a wind generator using the ANSYS FLUENT software package // Technical Physics: Pleiades Publishing. — 2016. — Vol. 61, No 2. — P. 299–301.

Н.К. Танашева, Э.Р. Шрагер, А.Н. Дюсембаева,
А.К. Кусаиынова, Д.А. Оспанова, А.Р. Бактыбекова

Қысымның таралуын анықтаған желтурбина қалақшаларын орап ағудың көрнекілік моделі

Мақала 3D кеңістіктегі желқозғалтқыш қалақшаларының аэродинамикасын модельдеуді үйренуге арналған. Желтурбина қалақшасында орналасқан шексіз цилиндр түріндегі ауа кеңістігінің торлы моделі құрылды. Модельдеу ауа ағынының әртүрлі жылдамдығы кезіндегі желкенді қалақшаны көлденен орап ағудың суретін алуға мүмкіндік берді. Ауа ағынының жылдамдығы 5 м/с тең кездегі α=0° бұрылу бұрышындағы симметрия жазықтығының ток сызығы мен қысымының таралуы анықталды.

Н.К. Танашева, Э.Р. Шрагер, А.Н. Дюсембаева,
А.К. Кусаиынова, Д.А. Оспанова, А.Р. Бактыбекова

Иллюстрационное моделирование обтекания лопасти ветротурбины с выявлением распределения давления

Статья посвящена изучению моделирования аэродинамики лопасти ветродвигателя в 3D пространстве. Построена сеточная модель воздушного пространства в виде бесконечного цилиндра, в которой расположена лопасть ветротурбины. Моделирование позволило получить картину поперечного обтекания парусной лопасти при различных скоростях потока воздуха. Получена картина распределения давления и линии тока в плоскости симметрии для угла атаки α=0° при скорости набегающего потока, равной 5 м/с.