CFD MODELING OF AERODYNAMIC FLOW IN A WIND TURBINE WITH VERTICAL ROTATIONAL AXIS AND WIND FLOW CONCENTRATOR
/
CFD МОДЕЛЮВАННЯ АЕРОДИНАМІЧНОГО ПОТОКУ У ВІТРЯНІЙ ТУРБІНІ З ВЕРТИКАЛЬНОЮ ВІССЮ ОБЕРТАННЯ І КОНЦЕНТРАТОРОМ ВІТРОВОГО ПОТОКУ

Gorobets V.G.[1)], Trokhaniak V.I.[1)], Masiuk M.Yu.[1)], Spodyniuk N.A.[1)], Blesnyuk O.V.[2)], Marchishina Ye.I.[1)] [1]

[1)] National University of Life and Environmental Sciences of Ukraine / Ukraine;

[2)] Kharkiv Petro Vasylenko National Technical University of Agriculture / Ukraine.

Tel: +380673513082; E-mail: Trohaniak.v@gmail.com

DOI: 10.35633/INMATEH-64-15

Keywords: *wind turbine, CFD, wind flow concentrator, speed, pressure*

ABSTRACT

In order to improve the aerodynamic characteristics by increasing the energy of the wind flow of wind turbines with a vertical rotational axis, a special device - wind flow concentrator was proposed. The concentrator contains channels with a curved contour, which are installed around the rotor. To study the hydrodynamic parameters of the wind flows concentrator depending on the ratio between the input and output geometric characteristics of the channels, as well as the angles of entry and exit methods of mathematical modeling are used. Based on the method of quadratic rotational design of orthogonal combinations, using three-dimensional numerical simulation, the aerodynamic characteristics of the wind flow concentrator for a vertical axis wind turbine were investigated.

РЕЗЮМЕ

Для покращення аеродинамічних характеристик підвищення енергії вітрового потоку вітротурбін з вертикальною віссю обертання було запропоновано спеціальний пристрій «концентратор вітрових потоків» з каналами, що мають криволінійний контур, які встановлюються навколо ротора. Методами математичного моделювання проведені дослідження гідродинамічних параметрів концентратора вітрових потоків в залежності від співвідношення між вхідними і вихідними геометричними характеристиками каналів, а також кутами входу та виходу. На основі методу квадратичного поворотного проектування ортогональних комбінацій, за допомогою тривимірного чисельного моделювання були досліджені аеродинамічні характеристики концентратора вітрових потоків для вертикально-осьової вітроелектричної установки.

INTRODUCTION

At present, in energy production is used the combustion of coal, crude oil and natural gas refining products. However, gradual depletion of natural resources, their rise in price, along with climate change, which is caused by harmful emissions from traditional energy sources have become major problems for countries around the world. Because of this problem, the main efforts of energy sector are directed at transition to new energy production technologies.

Most countries have committed themselves to reducing harmful emissions into environment under the United Nations Framework Convention on Climate Change and the Kyoto Protocol. Therefore, one of the most important energy problems is the development and implementation of energy efficient and safe technologies for obtaining heat and electricity. These technologies include the development of energy devices that use environmental energy - solar and wind energy, soil energy, and so on. These devices are called alternative or renewable energy sources.

---

[1] *Gorobets V.G., Head of Dep. DSc. Eng.; Trokhaniak V.I., Assoc. Prof. Ph.D. Eng.; Masiuk M.Yu., Ph.D. Student; Spodyniuk N.A., Assoc. Prof. Ph.D. Eng.; Blesnyuk O.V., Assoc. Prof. Ph.D. Eng.; Marchishina Ye.I., Assoc. Prof. Ph.D. Eng.*

One of the directions of progress in renewable sources is the development of devices that use wind energy - wind power plants. Based on design features of the wind turbine, there are installations with a horizontal rotational axis of the rotor and vertical axis wind turbines. The most widespread are wind turbines with a horizontal axis of the flow, which have a large capacity. One of the disadvantages of these plants, unlike vertical axis wind turbines, is the low efficiency at small wind speeds.

In the design of vertical axis wind turbine two types of rotors have been proposed: Savonius and Darya rotors, differing in the shape of the blades used in them. Further, the main efforts of researchers were focused on the rotors development with improved hydrodynamic characteristics. In particular, the authors of *Seki K. (1981)* developed a special profile of the wing of a vertical axis wind turbine, which was used on an operating wind turbine in Japan. A number of researchers have changed the shape of the aerodynamic profile of the blades, which would improve their starting characteristics at low wind speeds (*Zamani M. et al., 2016*). Studies of straight-bladed vertical axis wind turbine were carried out. In addition to changing the profile of the wing, they used, in particular, a wavy leading edge, a wavy trailing edge, and a flexible blade (*Favier J. et al. 2012; Zhenyu W. et al., 2018*). However, given the difficulties of practical manufacture of this type of blades, they need further improvement.

Another way to improve characteristics of a vertical axis wind turbine is to change the traditional design of straight blades in order to improve their performance and reduce aerodynamic resistance at low and high wind speeds. In particular, *Abdalrahman G. et al., (2017),* studied the effect of changing the pitch angle of the blades on the efficiency of wind turbines at different values of wind speeds. An analysis of the forces acting on the blade, which has a different pitch angle at the same speed of the blade (tip) edge during rotation was performed. The law of variation of rotational speed of the blades depending on the blade pitch angle was obtained.

Installation of wind flow concentrators on the rotor has been the most common area of research in recent years, which was mainly to improve the conditions of self-starting of the rotor. *Ji J.F., et al., (2012)*, proposed a variety of vertical axis wind turbine with wind shields, mounted around a vertical axis wind turbine with straight blades. These wind shields direct the incoming wind flow straight to the surface of the blades, which increases the rotational speed of the rotor. The research results have shown that these devices make it possible to improve the starting characteristics and power of wind turbines. Similarly, *Xiaohang W. et al. (2018)* patented a V-shaped orienting screen for power devices, generating electricity with the simultaneous use of solar and wind energy. Experimental studies have shown that self-starting characteristics and rotational speed of the blades of the vertical axis wind power plant when using V-shaped orienting screens were much better compared to a vertical axis wind power plant without an orienting screen.

Wong K.H. et al., (2018) studied the aerodynamic characteristics and the field of wind flow velocities using narrowing screens, which were located in the lower and upper part of the wind turbine. The use of this type of screens increased the wind flow velocity, as well as reduced the initial speed of the turbine.

The paper published by *Li Q. et al., (2021)* investigated the evaluation of the aerodynamic performance and vortex characteristic of a straight-bladed Vertical Axis Wind Turbine (VAWT) in the spanwise direction. To observe the aerodynamic forces characteristics, pressures acting on the blade surface were measured by a multiport pressure measurement device. Pressure measurement was carried out in the spanwise sections of $z/(H/2)$ = 0, 0.4, 0.70, 0.8 and 0.90 with wind tunnel experiments. This study provided a better understanding of the development of aerodynamic forces and vortex characteristic through pressure measurements and panel method calculations.

In this study, a conceptual co-axial contra-rotating vertical axis wind turbine (CR-VAWT) is conceived by splitting the single conventional vertical axis wind turbine blade length into two equal lengths forming two rotors separated by a small distance about the same axis of rotation. The available literature lacks much systematic investigation to find the best possible numerical setting and to extract an in-depth understanding of dominant factors limiting the aerodynamic performance of the CR-VAWT using CFD, which constitutes the primary focus for the present study. The present adopted concept of CR-VAWT is found to have negligible aerodynamic torque and side-side force action at the tower base (*Poguluri S.K. et al., 2021*).

The paper *Orlando A. et al., (2021)* discusses the response and fatigue damage of the supporting tower of a small vertical axis wind turbine subject to stationary and non-stationary excitation due to wind, turbine rotation, emergency stop and start. The emphasis of this paper is put on two main aspects: the use of full scale data and the detection of isolated events of non-stationary vibrations, the effects of which can be disregarded when using stress time-histories of 10 min - 1 h.

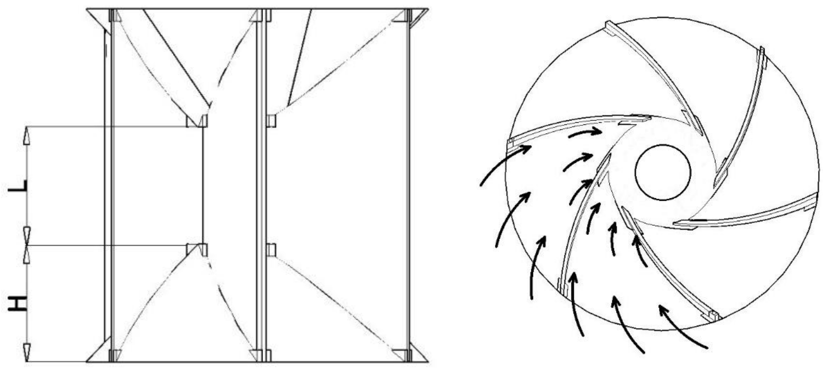
The floating vertical axis wind turbine (VAWT) is considered as a competitive device in the utilization of offshore wind energy. However, the platform pitch motion would affect its aerodynamic behavior. In this paper, the aerodynamic performance of a floating φ-type VAWT under pitch motion is investigated by using the Improved Delayed Detached Eddy Simulation SST turbulence model. The results showed that the averaged net power coefficient increment of about 1.5%–15% could be obtained under platform pitch motions, and the fluctuation of aerodynamic loads was found to increase (*Su J. et al., 2021*).

Authors *Hohman T.C. et al., (2020)* present the results of an experimental study of the effects of blade sweep on the wake characteristics of VAWTs performed using high resolution particle image velocimetry. Several sweep configurations were tested while holding the turbine solidity constant. As the sweep angle was increased, dynamic stall was reduced, which could yield a potential reduction in the amplitude of cyclical torque variations. Increased sweep also resulted in a more uniform wake exhibiting a faster wake recovery, and an increase in the planform energy flux in the turbine wake when compared to the straight-bladed turbine. The results support previous suggestions that the power generation of VAWT farms is bounded by the planform energy flux from above.

Authors of *Ipate G. et al., (2020)* worked on the numerical study of the effects of the fluid interaction with the blades of the working bodies of the renewable energy conversion systems. The paper presents the results of numerical research regarding the influence of the blade shape (straight), the number of blades (2, 3, 4) or the wind speed in the area on the output power of the small capacity wind turbines.

MATERIALS AND METHODS

One of the best designs of the wind flow concentrator is the truncated-cone-shaped wind gathering device with orienting channels, which is shown in Fig. 1. The wind flows concentrator consists of confuser-type narrowing channels system. The upper and lower parts of these channels are closed by conical surfaces, which make it possible to concentrate the wind flow in the direction of the rotor. According to the equation of flow continuity in incompressible liquid (gas) (*Li Y. et al., 2018*), the use of confuser-type narrowing channels increases the flow velocity and improves the concentration of wind flow energy in the direction of the rotor. The geometry of channel orienting shoulder blades should be chosen to direct the wind flow at optimal angle of attack on the turbine blade. The proposed design of wind flow concentrator is universal and works regardless of the wind direction, which flows through the orienting channels of confuser-type, where its speed increases due to Venturi effect. As a result of using this type of device it is possible to get more wind flow power per unit area for any wind direction, which increases the wind turbine productivity. Given the fact that electromagnetic moment and power of a wind turbine are proportional to the square and cube of the wind speed, respectively, the use of a wind flow concentrator will improve the aerodynamic characteristics of a vertical axis wind turbine.



a) b)

Fig. 1 – Appearance of the wind flow concentrator
a) side view, b) sectional view (top)

Further, experimental studies of the wind flow concentrator will be carried out in a wind tunnel, which has the following characteristics - the size of the working area of 0.3×0.3 m, the range of changes in wind speed in the working area from 0 to 18 m/s. Based on these characteristics, a model of the rotor of wind flow concentrator was selected and designed with the following parameters, which are shown in table 1.

Structural parameters of the wind flow concentrator **Table 1**

Value	Dimensionality	Quantity
Cone height *H*	m	0.1
The height of the working area *L*	m	0.1
The thickness of the blade *l*	m	0.002
The size of the working area *L*x*R*	m	0.1x0.1
The area of the inlet of the channel S_1	m^2	0.0471
The area of the outlet of the channel S_2	m^2	0.0052

The general concept of the wind flow concentrator is to increase the energy of wind flow per unit area, increasing torque over the entire area of the blades, reducing the turbulence of the oncoming flow and stabilizing the rotor speed. The characteristics of the wind flow concentrator can be found by modeling the aerodynamic flows using CFD modeling included in the commercial package ANSYS Fluent.

Modeling of hydrodynamic flows is based on the solution of three-dimensional Navier-Stokes equations (*Gorobets V.G. et al., 2018; Khmelnik S.I., 2018;*), in which the laws of conservation of mass and momentum are formulated. A standard k-ε turbulence model was used (ANSYS, 2017; *Gorobets V.G. et al., 2018*). The computational area of the design of the wind flow concentrator is divided into cells. The numerical algorithm for solving the Navier-Stokes equations is based on use of the finite volume method, and the calculation of equations is performed for each cell in the computational domain. ANSYS Meshing software was used to create a three-dimensional grid for the computational model.

The Navier-Stokes equations for the turbulent flow of an incompressible liquid (gas) are as follows:

$$\left.\begin{aligned}
\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)&=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right),\\
\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)&=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right),\\
\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)&=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)
\end{aligned}\right\}$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0. \qquad (1)$$

The following boundary conditions are set on the walls of the channels and at its entrance and exit:

$$\begin{aligned}
&u(t=0)=v(t=0)=w(t=0)=0;\\
&at \quad x=x_{wall},\ y=y_{wall};\ z=z_{wall};\quad u=v=w=0,\ \frac{\partial u_{wall}}{\partial \bar{n}}=\frac{dv_{wall}}{\partial \bar{n}}=\frac{dw_{wall}}{\partial \bar{n}}=0,\\
&at \quad x=x_{ent};\ y=y_{ent};\ z=z_{ent};\quad u_{ent}=v_{ent}=w_{wall}=W_0;\\
&at \quad x=x_{exit};\ y=y_{exit};\ z=z_{exit};\quad \frac{\partial u_{exit}}{\partial x}=\frac{\partial v_{exit}}{\partial y}=\frac{\partial w_{exit}}{\partial z}=0.
\end{aligned} \qquad (2)$$

For the given model, the following designations are accepted: *x,y,z* – Cartesian coordinates; *t* – time; *u,v,w* – speed components; *ρ* – air density; $\bar{n}$ –normal vector on the walls of the channels; the indices wall denote the walls of the channels; *ent, exit* – entrance and exit of channels; W_0 – air speed at the entrance to the channels.

Given the three-dimensionality of hydrodynamic air flows, it is necessary to consider a three-dimensional calculation model. The computational area for the wind flow concentrator is shown in Fig.2. The center of the counting system is the center of the rotor, and the direction of wind flow runs along the axis *0x*. The central rotational axis of the wind turbine is the axis *0z*. The length of the computational domain was taken equal to 1500 mm in the flow direction, and the cross-sectional area was taken equal to 290x290 mm, corresponding to the size of the wind flow concentrator, which will be used in the future for experimental studies in the wind tunnel. At the entrance to the computational domain, the wind flow velocity is constant and equal to 1 m/s. At the exit of the domain was set the value of the pressure, which is equal to atmospheric pressure. Standard conditions for flow adhesion are set on the side, upper and lower surfaces of the wind flow concentrator.

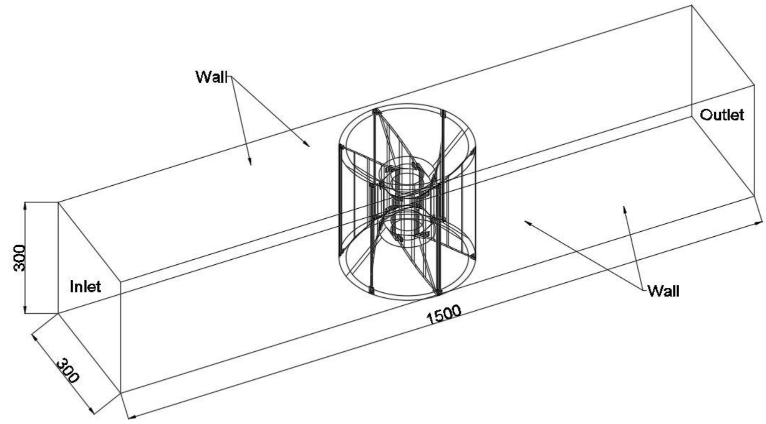


Fig. 2 – Computational domain scheme

In the computational model, a tetragonal unstructured grid was set, which is shown in Fig. 3. This type of grid has the greatest adaptability and high accuracy for modeling a wind turbine.

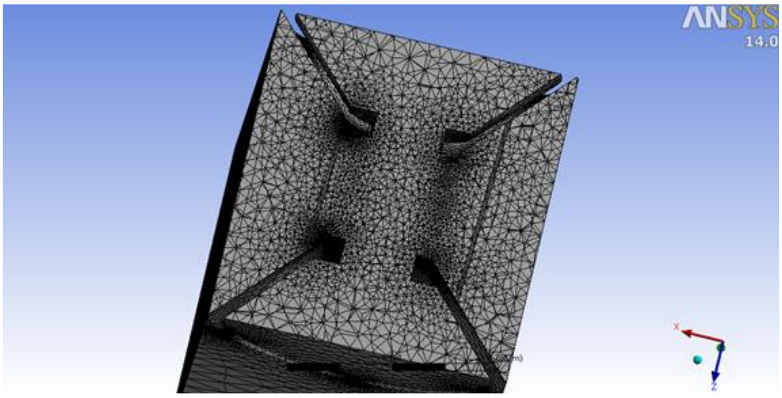


Fig. 3 – Estimated mesh

RESULTS

As a result of the numerical simulation in the wind flow concentrator, the velocity field for the cross section along the axis 0z and the longitudinal section of the axis 0y was obtained (see Fig. 4).

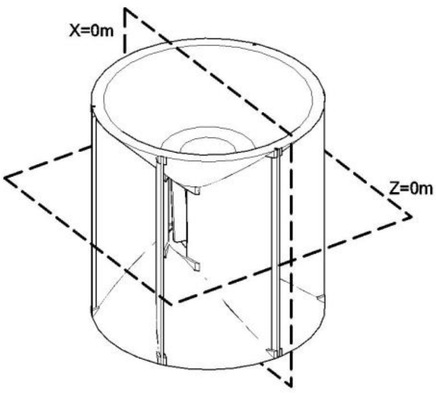
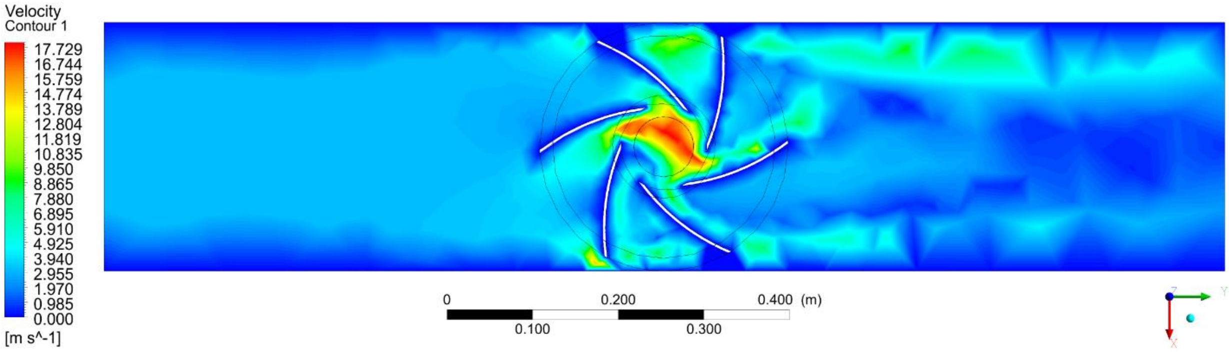
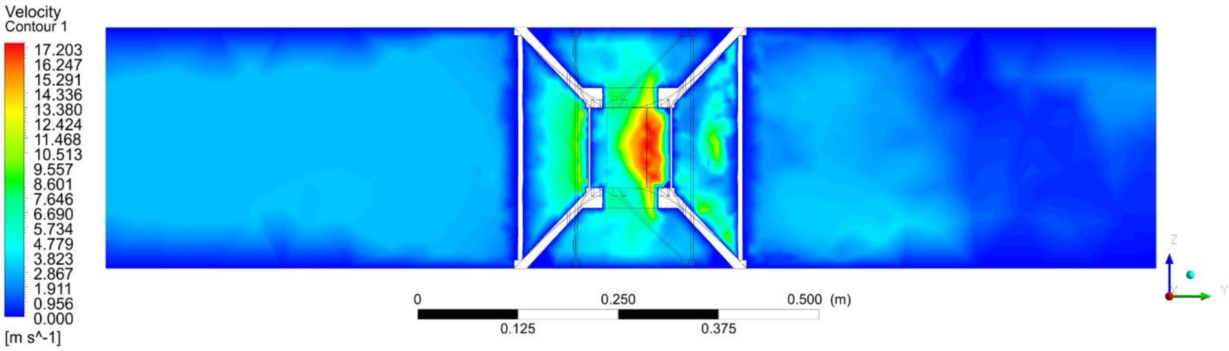


Fig. 4 – The location of the cross section in the numerical calculation model

The results obtained on the basis of numerical modeling of hydrodynamic processes in the inner part of the wind flow concentrator for the velocity field in the cross section of the working area are presented in Fig. 5-6. As can be seen from the figure, a significant increase in air velocity with maximum values at certain points up to 18 m/s is observed in the inner part of the wind flow concentrator. The average air velocity in the inner part of the wind flow concentrator is about 11 m/s. At the exit of the wind flow concentrator, increased turbulence can be observed (Fig. 6).

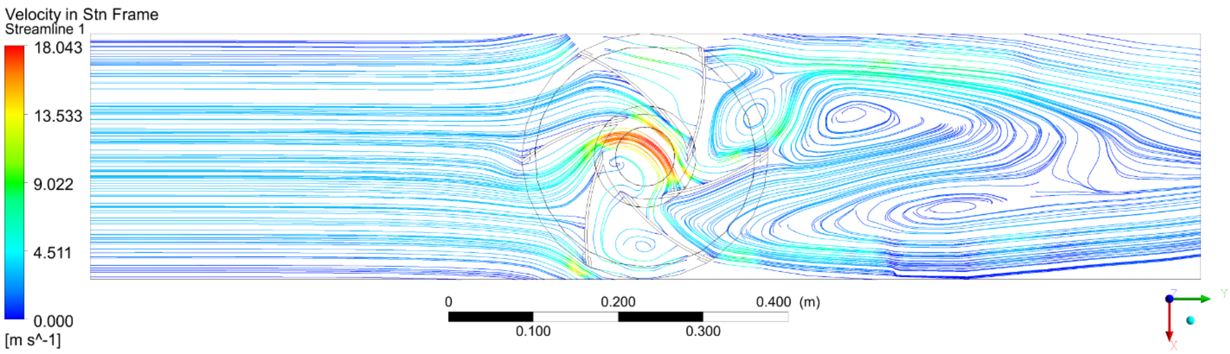


a)

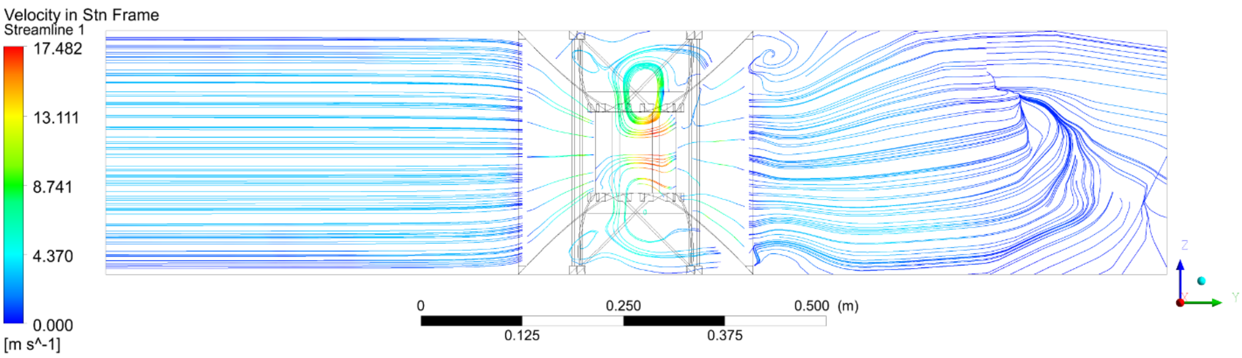


b)

Fig. 5 – The velocity field in the cross section along the axis *0z* (a) and the longitudinal section of the axis *0x* (b) of the wind flow concentrator, m/s



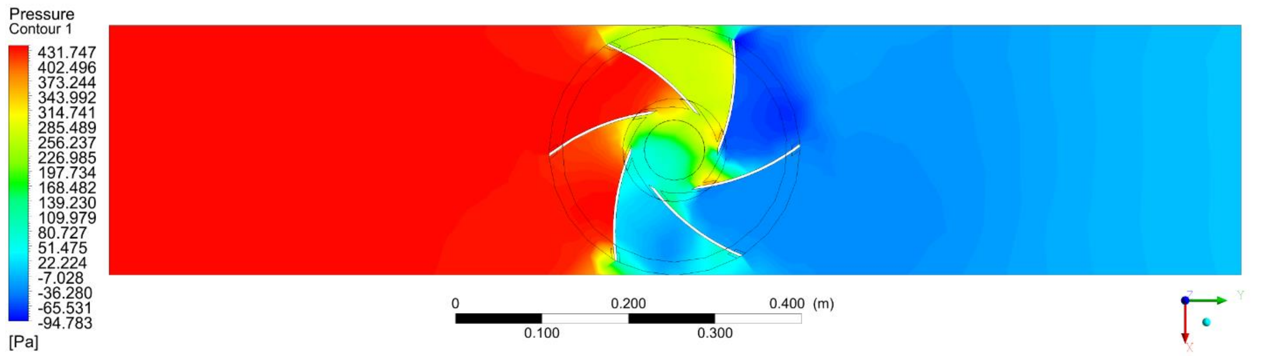
a)



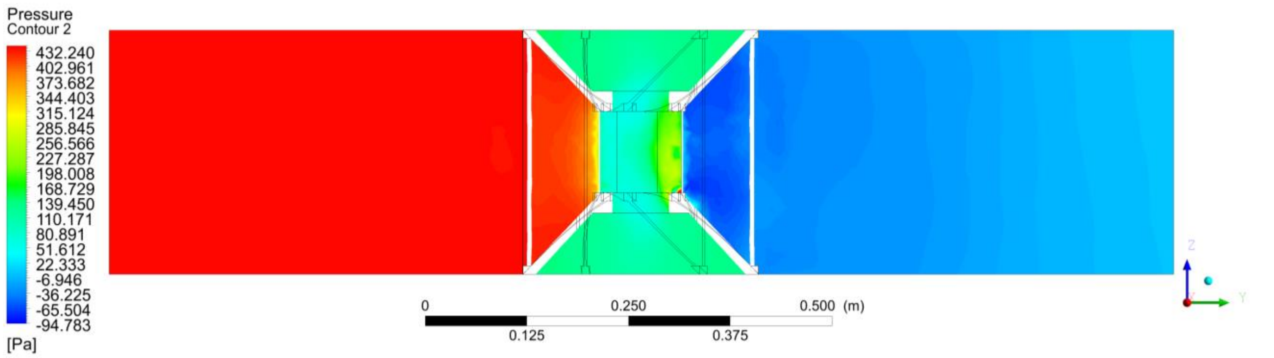
b)

Fig. 6 – Current lines in the cross section along the axis *0z* (a) and the longitudinal section of the axis *0x* (b) of the wind flow concentrator, m/s

The pressure drop is insignificant and reaches 432 Pa (Fig. 7).



a)



b)

Fig.7 – Pressure loss in the cross section along the axis *0z* (a) and the longitudinal section of the axis *0x* (b) of the wind flow concentrator, Pa

Analysis of the results of numerical modeling of hydrodynamics processes in wind flow concentrator shows that the use of these devices leads to increase the wind flow energy in the cross-section per unit cross-sectional area, as well as to raise the torque, which leads to the increase in rotor speed. Application of proposed design of the wind flow concentrator makes it possible to increase the productivity of vertical axis wind turbines and use them at low wind speeds.

CONCLUSIONS

1. An innovative device is presented - a wind flow concentrator which can be used to increase the productivity of a vertical axis wind turbine. The wind flow concentrator was designed to increase the energy of the oncoming wind flow, to raise the rotor torque and the use of wind flow energy in a wind turbine at low wind speeds. The wind flow concentrator is universal and works regardless of the wind direction, which flows through the orienting channels of confuser-type, where its speed increases due to Venturi effect.

2. CFD modeling of aerodynamic flows using the commercial package ANSYS FLUENT was performed. The velocity fields and pressure changes in the investigated design of the wind flow concentrator are obtained. In the inner part of the wind flow concentrator there is a significant increase in air velocity, which increases the rotational speed of the rotor.

3. The proposed design of the wind flow concentrator can be used in a vertical axis wind turbine in order to increase its productivity. Using a wind flow concentrator with a vertical axis wind turbine, in contrast to a wind turbine with a horizontal rotational axis of the rotor, makes it possible to apply wind turbines in areas with low wind speeds.

REFERENCES

[1] Abdalrahman G., Melek W., Lien F.S., (2017), Pitch Angle Control for Small-Scale Darrieus Vertical Axis Wind Turbine with Straight Blades (H-type VAWT), *Renewable Energy,* vol. 114, pp. 1353-1362, https://doi.org/10.1016/j.renene.2017.07.068, United Kingdom;

[2] ANSYS, (2017), *ANSYS Fluent Theory Guide. Release 18.2*, Published in the USA, 832 p.;

[3] Favier J., Pinelli A., Piomelli U., (2012), Control of the separated flow around an airfoil using a wavy leading edge inspired by humpback whale flippers, *Comptes Rendus Mécanique*, vol. 340, pp. 107-114, https://doi.org/10.1016/j.crme.2011.11.004, Paris/France;

[4] Gorobets V.G., Bohdan Yu.O., Trokhaniak V.I., Antypov I.O., (2018), Experimental studies and numerical modelling of heat and mass transfer process in shell-and-tube heat exchangers with compact arrangements of tube bundles, *MATEC Web of Conferences*, vol. 240, 02006, https://doi.org/10.1051/matecconf/201824002006, Cracow/Poland;

[5] Gorobets V.G., Trokhaniak V.I., Antypov I.O., Bohdan Yu.O., (2018), The numerical simulation of heat and mass transfer processes in tunneling air ventilation system in poultry houses, *INMATEH: Agricultural Engineering*, vol. 55, no. 2, pp. 87-96, Bucharest/Romania;

[6] Hohman T.C., Martinelli L., Smits A.J., (2020), The effect of blade geometry on the structure of vertical axis wind turbine wakes, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 207, 104328, https://doi.org/10.1016/j.jweia.2020.104328, Netherlands;

[7] Ipate G., Constantin G.A., Biris S.S., Voicu G., Rusanescu C.O., Stefan V., (2020), Numerical 3D analysis of a mini wind turbine with horizontal axis, for implementation in agricultural farms, *INMATEH: Agricultural Engineering*, vol. 63, no. 2, pp. 325-332, https://doi.org/10.35633/inmateh-62-34, Bucharest/Romania;

[8] Ji J.F., Deng Z.Y., Jiang L., (2012), Optimization design of a 5 kW lift type vertical axis wind turbine with wind shield-growth patterns, *Journal Engineering Thermophysics*, vol. 33, no. 7, pp. 560-564, Russian Federation;

[9] Khmelnik S.I., (2018), Navier-Stokes equations. On the existence and the search method for global solutions, *Mathematics in Computers – MiC*, 134 p., Bene-Ayish/Israel;

[10] Li Q., Cai C., Maeda T., Kamada Y., Shimizu K., Dong Y., Zhang F., Xu J., (2021), Visualization of aerodynamic forces and flow field on a straight-bladed vertical axis wind turbine by wind tunnel experiments and panel method, *Energy,* vol. 225, 120274, https://doi.org/10.1016/j.energy.2021.120274, United Kingdom;

[11] Li Y., Zhao S., Tagawa K., Feng F., (2018), Starting performance effect of a truncated cone-shaped wind gathering device on small-scale straight-bladed vertical axis wind turbine, *Energy Conversion and Management*, vol. 167, pp. 70-80, https://doi.org/10.1016/j.enconman.2018.04.062, United Kingdom;

[12] Orlando A., Pagnini L., Repetto M.P., (2021), Structural response and fatigue assessment of a small vertical axis wind turbine under stationary and non-stationary excitation, *Renewable Energy,* vol. 170, pp. 251-266. https://doi.org/10.1016/j.renene.2021.01.123, United Kingdom;

[13] Poguluri S.K., Lee H., Bae Y.H., (2021), An investigation on the aerodynamic performance of a co-axial contra-rotating vertical-axis wind turbine, *Energy*, vol. 219, 119547, https://doi.org/10.1016/j.energy.2020.119547, United Kingdom;

[14] Seki K., (1981), Vertical axis wind turbine designed aerodynamically at Tokai University, *Periodica Polytechnic Mechanical Engineering,* vol. 25, no. 1, pp 47-56, Budapest/Hungary;

[15] Su J., Li Y., Chen Y., Han Z., Zhou D., Zhao Y., Bao Y., (2021), Aerodynamic performance assessment of φ-type vertical axis wind turbine under pitch motion, *Energy*, vol. 225, 120202, https://doi.org/10.1016/j.energy.2021.120202, United Kingdom;

[16] Trokhaniak V., Klendii O., (2018), Numerical simulation of hydrodynamic and heat-mass exchange processes of a microclimate control system in an industrial greenhouse. *Bulletin of the Transilvania University of Brasov, Series II: Forestry, Wood Industry, Agricultural Food Engineering*, vol. 11 (60), no. 2, pp. 171-184, Brasov/Romania;

[17] Wong K.H., Chong W.T., Sukiman N.L. (2018), Experimental and simulation investigation into the effects of a flat plate deflector on vertical axis wind turbine, *Energy Conversion and Management*, vol. 160, pp. 109-125, https://doi.org/10.1016/j.enconman.2018.01.029, United Kingdom;

[18] Xiaohang W., Wentong C., Kokhoe W., Liphuat S., Sinchew P., Saihin L., Chin-Tsan W., (2018), Preliminary Performance Tests and Simulation of a V-Shape Roof Guide Vane Mounted on an Eco-Roof System, *Energies*, vol. 11, no. 10, 2846; https://doi.org/10.3390/en11102846, Basel/Switzerland;

[19] Zamani M., Javad M.M., Rasoul V.S., (2016), Starting torque improvement using J-shaped straight-bladed Darrieus vertical axis wind turbine by means of numerical simulation, *Renewable Energy*, vol. 95, pp. 109–126, https://doi.org/10.1016/j.renene.2016.03.069, United Kingdom;

[20] Zhenyu W., Yuchen W., Mei Z., (2018), Improvement of the aerodynamic per blades using CFD and Taguchi method, *Energy Conversion and Management*, vol. 177, pp. 107-121, http://dx.doi.org/10.1016/j.enconman.2018.09.028, United Kingdom.