UDC 621.548 (045)

[1]V. M. Sineglazov,
[2]V. M. Boyko,
[3]A. V. Kulbaka

INTEGRATED COMPUTER-AIDED DESIGN SYSTEM OF WIND-POWER PLANT

Aviation Computer-Integrated Complexes Department, National Aviation University, Kyiv, Ukraine
E-mails: [1]svm@nau.edu.ua, [2]detroit324@ukr.net, [3]antonkul@ukr.net

Abstract—*It is proposed a power plant represented a combination of turbo rotors: Darrieus and Savonius. It is showed that the optimal construction of this power plant is possible only by use the computer-aided design system. The structure scheme of computer-aided design system is developed. Rotor tested in wind tube.*

Index Terms—Computer-aided design; wind power plant; rotor modeling

I. INTRODUCTION

In simulation of wind turbines the very important role wins back the simulation of model to determine the power and profitability of the future wind turbines before spending resources on construction of the real model. In this case it is possible to choose one of the options:

– construction of the reduced model;

– design and calculation of the models in the software.

Advantages of the second option are the lowest price and the ability to change the parameters of the model at any moment, that is the versatility of the method.

The software FlowVision may be suitable for the construction.

Commercial code FlowVision is intended for modeling of three-dimensional gas and fluid flows in technical and natural objects with subsequent visualization of the results using different methods of computer graphics. The modeled flows may be steady or unsteady, laminar or turbulent, compressible, weakly compressible, or incompressible. FlowVision is based on the finite-volume approach to the integration of the fluid dynamics equations. It uses a hexahedral adaptive grid with local refinement. Sub-grid resolution technology is implemented for accurate approximation of curvilnear geometry. The technology enables user to import geometries from computer-aided design (CAD) systems and to interact with finite-element systems. The technology implies automatic grid generation, viz., only several parameters are to be specified in order to build the grid for a computational domain of arbitrary complexity. Aces to the model parameters and posibilty to locally adapt computational grid alow simulation of complex flows with strong swirl, fre surfaces, combustion, etc. To start working with FlowVision, you have to install Windows ME/200/XP and a CAD system for creation of the geometry of computational domain on your PC. The following CAD systems are recommended: SolidWorks, T-Flex, Unigraphics, Autocad Mechanical Desktop, ProEnginer, Catia. In this work we use the SolidWorks to simulate the geometry.

Calculation of a fluid flow implies the following steps:

– creation of the geometry of computational domain in a CAD system and import of the geometry to FlowVision from the VRML, STL, DEFORM, ABAQUS, ANSYS or NASTRAN (IGES, PARASOLID, VDAFS by using Flow3DVision);

– specifying mathematical model;

– setting boundary conditions;

– specifying primary grid and criteria for its adaptation to the solution and boundaries;

– specifying parameters of the numerical methods used for integration of different equations;

– calculations, which are carried out without user interference;

– visualization of the obtained results, saving the needed characteristics in files.

Repetition of calculations on a finer grid in order to estimate the accuracy of the results [2].

II. STEPS OF CALCULATION

Step 1. Creating computational domain

The section describes how to create and import computational domain. Computational domain comprises the volume, where the governing equations are integrated, and the boundary of the volume, where the corresponding boundary conditions are set. The geometry of the computational domain is built in a CAD system outside FlowVision. The surfaces imported to FlowVision have to be closed sets of plane convex polygons, which are called 'facets' in the sequent. The surfaces are to be embeded in one another without intersection (Fig. 1).

Let N be the number of embeded surfaces S_1, S_2, …, S_n. They divide the three-dimensional space into N volumes V_1, V_1, …, V_n bounded by the surfaces. Volume V_0 is external to surface S_1. Any internal volume V_i can be a computational domain. No mathematical model can be assigned to the external volume V_0, i. e. it is never calculated [1].

http://ecs.org.ua

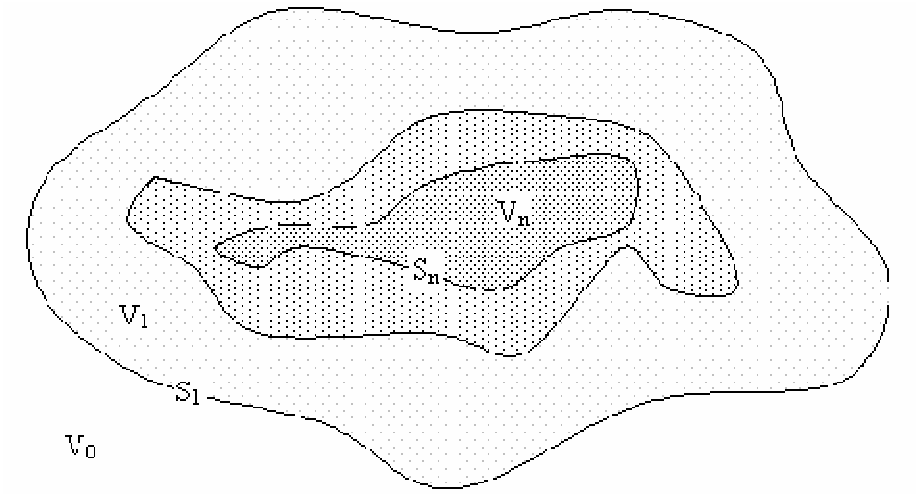


Fig. 1. Body surfaces

During the import of the geometry, FlowVision automatically creates the volumes and give default names SubRegion#1, SubRegion#2 etc. to them. Note that he numbers do not correspond to the order of the volume embedding. Since FlowVision does only three-dimensional calculations, a 3D domain has to be defined for the 2D problem discussed. You see the sketch of the domain in the first figure and the 3D domain in the second one. The domain is obtained by extrusion of the sketch in the direction normal to the sketch plane in our Darrieus rotor. The created geometry is saved as a .VRML or .STL file. In order to import it o FlowVision, select command New on menu File and chose the needed file in the dialog window. More details – se in volume User's guide. The next figure displays the computational domain imported to FlowVision. To create a construction you must connect the planes that will create a three-dimensional closed space. In this case, we can change construction elements of the rotor for maximum efficiency proportions and details. In this case, the classical model of Darrieus rotor was created with a height of 4 meters (Fig. 2a) and combined rotor with the same height (Fig. 2b).

This model is used to simulate power and verification of external influences on the rotor speed and, consequently, released energy during this.

Parameters of the tested rotor (Table 1).

Step 2. Choosing flow model

The section describes how to choose flow model and governing equations in each calculated sub-region. The goal of the flow simulation is to obtain the fields of velocity, pressure, and other quantities in the computational domain. The calculations require specifying the laws governing the flow (Fig. 3). The set of the laws constitute the mathematical model of the flow [2].

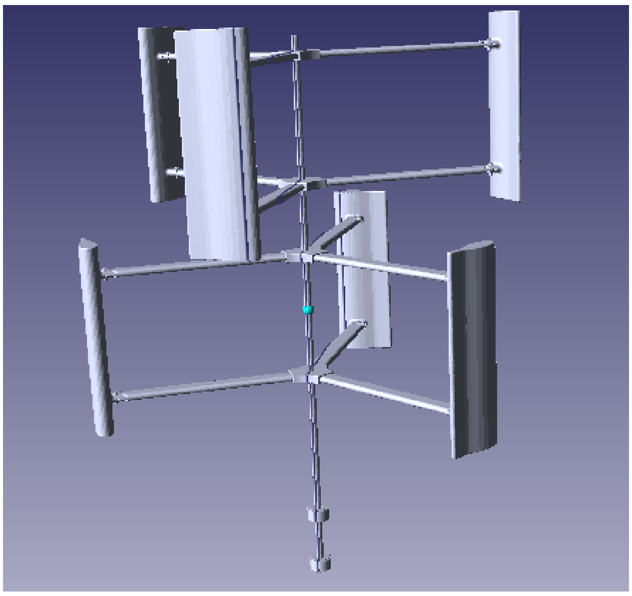
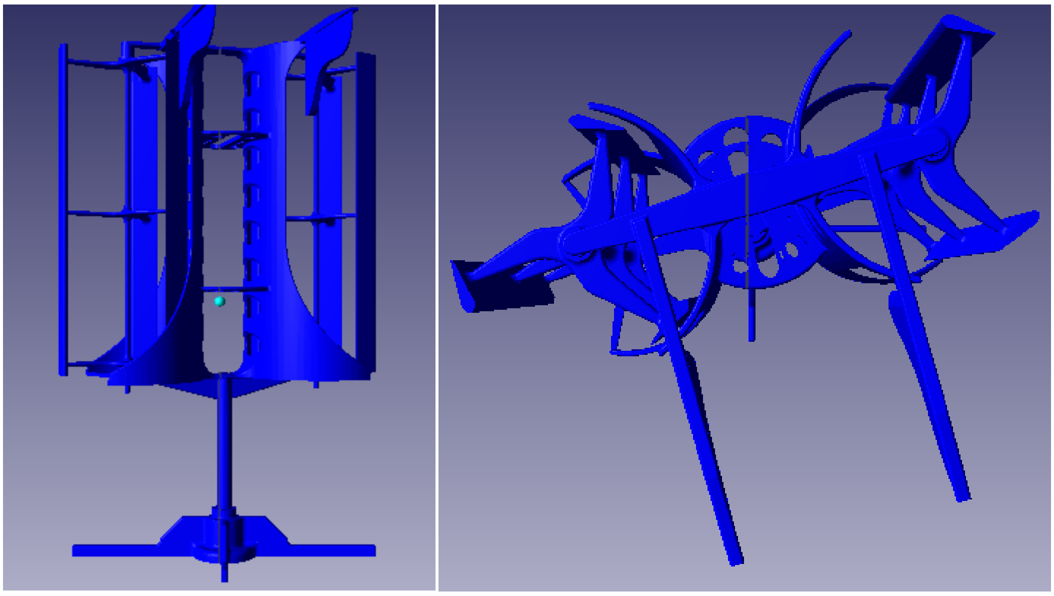


Fig. 2. Darrieus rotors



a b

Fig. 3. Combined plant: a is side view; b is top view

TABLE 1

PARAMETERS OF THE TESTED ROTOR

Parameters	Readings
Number of blades	6
Relative chord of the profile in relation to the radius of the blade	0.08
Nominal power	1000
Blade height	1.65
Wind generator height	4
Efficiency electrical	0.6
Efficiency mechanical	0.9

In general, the mathematical model is: a) a set of partial differential equations describing conservation of mass, momentum, and energy; b) corresponding boundary conditions; c) appropriate state equations. The model is specified and modified in the pre-processor.

A user chooses a model from the list, turns on and of the equations constituting the model, and changes the model parameters.

The computed variables are vectors and scalars. They are subdivided into independent and dependent ones [1].

The independent variables are computed from the governing equations, the dependent variables are expressed through the independent ones. For instance, velocity V is an independent vector variable, whereas its absolute value $\|V\|$ is a scalar dependent variable. Remind that the velocity field is determined by the Navier-Stokes and continuity equations.

The Table 2 demonstrates the correspondence between the equation names and variables [3].

TABLE 2

THE CORRESPONDENCE BETWEEN THE EQUATION NAMES AND VARIABLES

Name of equation/system	Independent variable	Notation	Dependent variable	Notation
Velocity	Velocity	V	Absolute value of velocity (ModVelocity)	$\|V\|$
	Pressure	P		
Temperature	Temperature	T		
Concentration	Concentration	C		
Turbulence	Turbulent energy (TurbEnergy)	k	Turbulent viscosity (TurbViscosity)	ν
	Dissipation rate of turbulent energy (TurbDissipation)	ε		
VOF	Volume fraction of fluid in a cell (VOF)	VOF		

The next step is to create a mathematical model of the object and choice of parameters to be calculated at compile input. In this case the object is a solid body surrounded by gas (air in this case). And we will compile several times with different parameters of air speed, angle of air flow that interacts with the rotor and the different pressures on the structure (to simulate the operation of the rotor in heavy rain or snow). To change the properties of the mathematical model parameters, such as viscosity, density of the liquid, the coefficients of density dependence on the concentration and temperature, you need to open a folder Subregion # 1, and then open the folder Physical specifications and to call the properties of the desired item, for example, the model parameters. In this case, it is possible to set both absolute and relative values. In this example we will use only the absolute parameter values (Figs 4, 5) [3].

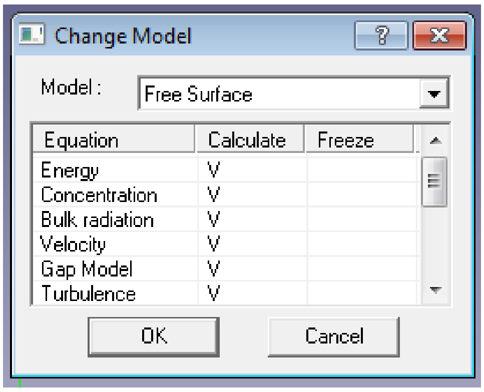


Fig. 4. Setting parameters

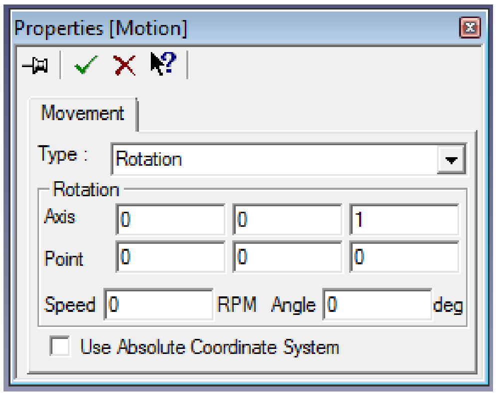


Fig. 5. Setting rotation

In this section you can set the properties of rotation of the body – point of rotation and the plane around which it will occur. As for influence of the rotating constructions on the flow of air we can specify rotational speed of construction. But in our case, it is the inverse problem, we leave the given fields blank [4].

Step 3. Setting the boundary conditions

Boundary conditions are to be specified on the boundary of the computational domain for each calculated variable. To make easy their choice and to avoid incompatible conditions, al the conditions are grouped under several types. Every Type of boundary corresponds to certain physical situation/process occurring on the boundary. The types, in turn, are united in two groups: 1. “Physical boundaries” and 2. “Special boundaries.”

The later group includes periodic, conjugate, and sliding conditions. The figure below shows the window for specifying/editing a boundary condition. To activate it, open folder BConditions in folder SubRegion#i, right click an existing boundary condition, and select command Properties on the pop-up menu. Click Ed. Chose the boundary type from list Type of Boundary and specify the appropriate boundary conditions for all the variables present in the window (Fig. 6) [6].

The third step is the defining boundary conditions for our wind wheel. In this step we define how the plane structures will interact with the air, which will rotate the rotor. For Darrieus rotor this task is simplified, because of the symmetry of the rotor and sameness of its blades. All the planes that constitute wind wheel will have the properties of the wall. This provides a maximum sampling coefficient of wind energy (Fig. 7) [4].

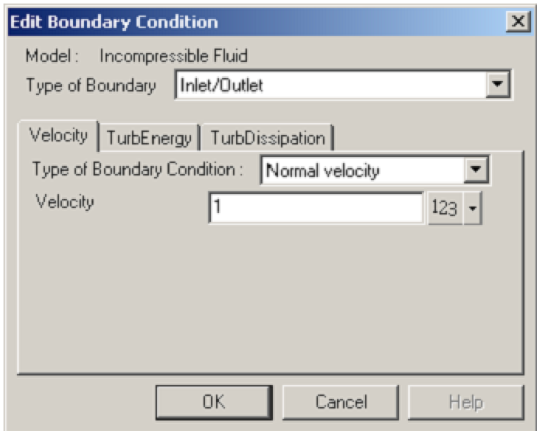


Fig. 6. Chosing of boundary conditions

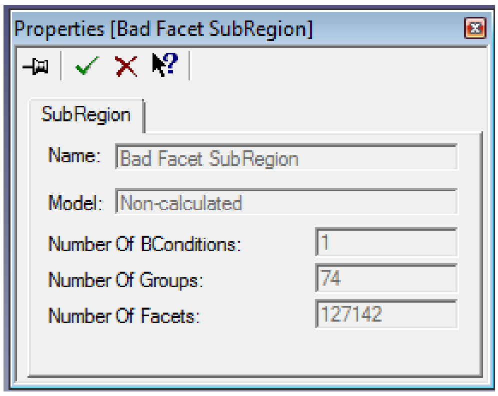


Fig. 7. Sub region

For our example we will not use the high class of accuracy, that will allow to simplify compilation of our model. So, we will not set a computational grid for detailing theconstruction elements and will define a second class of accuracy (Fig. 8) [3].

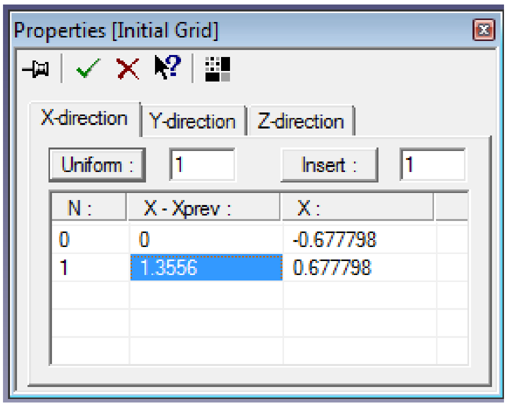


Fig. 8. Changing initial grid

Step 4. Generation of computational grid

FlowVision integrates the model governing equations on a hexahedral Cartesian adaptive grid with local refinement (HAGLR). Adaptation is applied when small details of the geometry or high gradients of the computed variables are to be resolved (Fig. 9) [2].

The essence of the HAGLR technology is as follows. The entire computational domain is covered with a hexahedral Cartesian mesh. In the regions, where peculiarities of the geometry or flow are to be resolved, every cell is divided into 8 equal smaller cells. If necessary, these new cells are divided further up to the required accuracy. In doing so, the cells of the initial mesh get index "0", the cells obtained by their division get index 1, and so forth. The condition is imposed that he indices of two cells contacting by faces or edges may differ by "1" maximum [3].

The method of sub-grid resolution of geometry implemented in FlowVision is intended for approximation of curvilnear boundaries on a hexahedral mesh. The essence of the method consists in the natural splitting of the boundary cells by the boundary facets. The number of the obtained child cells depends on the peculiarities of the geometry. The child cells are arbitrary polyhedrons. The equations of a given mathematical model are approximated on the polyhedrons without simplifications. The approach enables accurate calculations in a complex domain on a reasonably coarse mesh (Fig. 10).

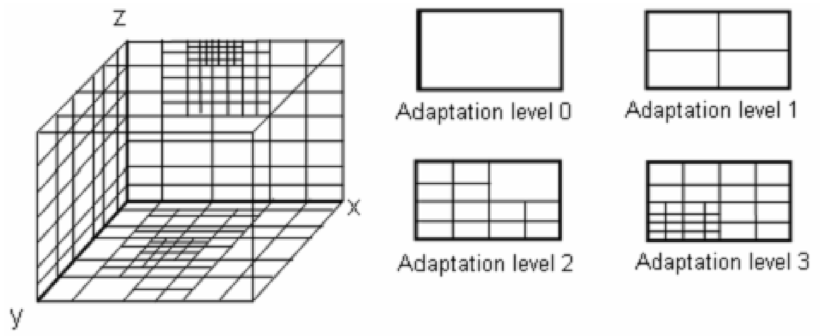
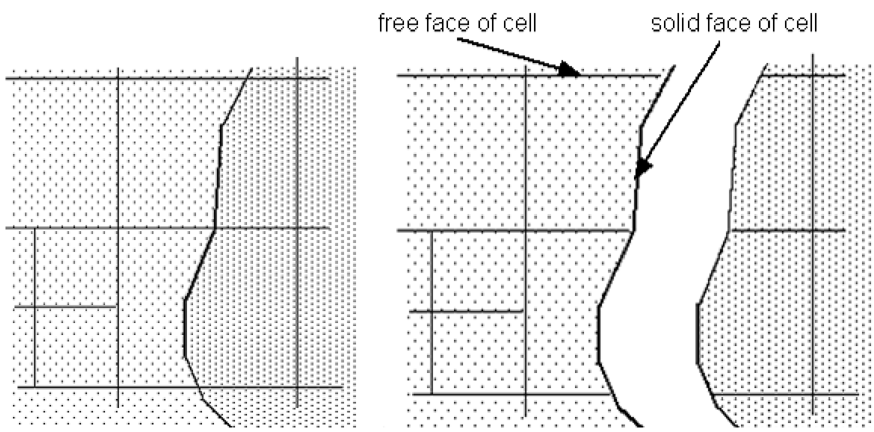


Fig. 9. Adaptive grid with local refinement



a b

Fig. 10. Method of sub-grid resolution of geometry: a is the boundary cells and facets; b is cell splitting

To specify the mesh of level "0" (the primary grid), right-click Initial Grid in the variant re and select command Properties on the pop-up menu. The three tabs serve to specify the grid along co-ordinates *X*, *Y*, and *Z*. The Figure 11 below illustrates how the primary grid is set for the flow around cylinder [4].

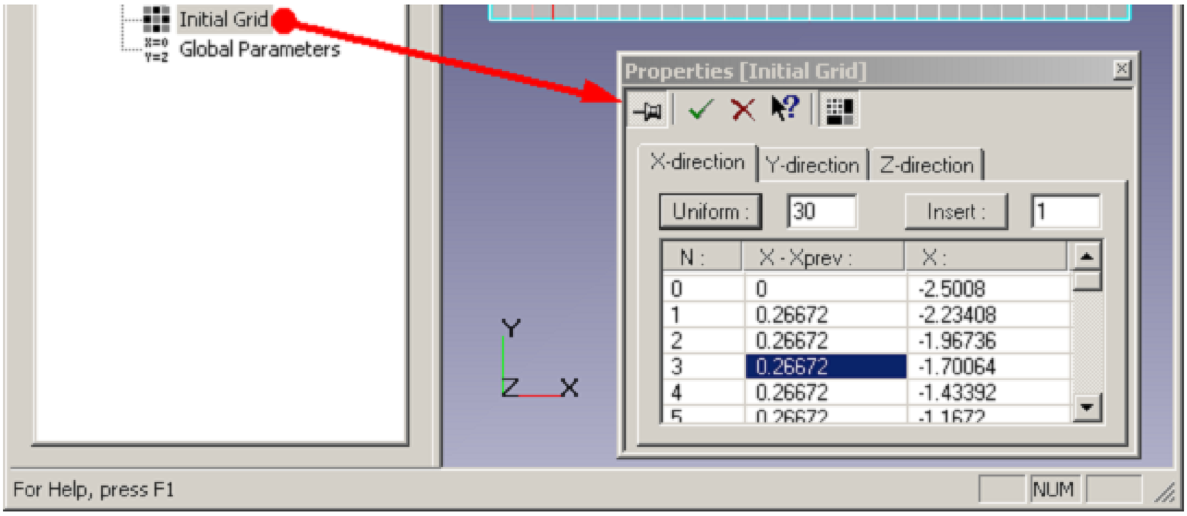


Fig. 11. Primary grid is set for the flow around cylinder

Step 5. Specifying numerical method

The section considers convection-diffusion equations and methods for their solving. Mathematical model describing a fluid/gas flow includes convection-diffusion equations [3]. Generalized differential form for such an equation is as follows:

$$\frac{\partial f}{\partial t} + \nabla(Vf) = \nabla(D\nabla f) + Q, \qquad (1)$$

where f is a calculated variable, V is the flow velocity, D is an appropriate diffusion coefficient, Q is the source term. FlowVision employs the finite-volume approach to numerical solution of the equation. Integrate Eq. (1) over the volume of cell i and over time interval t:

$$\int_{V_i} f dv \Big|_{t=t^{n+1}} - \int_{V_i} f dv \Big|_{t=t^{n}} + \int_{\tau} \oint_{S_i} f \mathbf{V} d\mathbf{s} dt = \int_{\tau} \oint_{S_i} D\nabla f d\mathbf{s} dt + \int_{\tau} \int_{V_i} Q dv dt, \qquad (2)$$

here V_i is the volume of the cell, S_i is its surface, t_n, $t_n + 1 = t_n + t$ are the terms corresponding to the beginning and end of the tome step. The cell is an arbitrary polyhedron. We shall say “free face” about a cell face contacting a neighbor cell. Denote the area of the free face j in cell i as j is. We shall say “solid face” about a cell face formed by the geometry boundary. Denote the area of the solid face j in cell i as j in g . Write Eq. (2) in finite-difference form:

$$V_i\left(f_i^{n+1} - f_i^n\right) + \sum_j F_i^j s_i^j + \sum_j G_i^j g_i^j + Q_i = 0, \qquad (3)$$

here Q_i is the volume source of variable f, f_n is its value obtained by averaging over the cell volume a time t_n: Fluxes j F_i and j G_i of variable f through the free and solid faces, obtained by integration over the time step, respectively are

$$V_i f_i^n = \int_{V_i} f dv \Big|_{t^n}. \qquad (4)$$

$$F_i^j = \int_{\tau} \left(f\mathbf{V} + D\nabla f\right) dt \Big|_{s_i^j}. \qquad (5)$$

$$G_i^j = \int_{\tau} \left(f_{wj}\mathbf{V}_{wj} + D\left(\nabla f\right)_{wj}\right) dt \Big|_{g_i^j}. \qquad (6)$$

Index w denotes the values of the quantities at the boundary. The second term in the integrand in Eq. (5) describes the diffusion flux of variable f. In FlowVision, it is approximated with the second order of accuracy in space [4]. The most complicated problem is approximation of the first term of the integrand in Eq. (5), Approximation of convection-diffusion equation. The term describes convective flux. Several schemes approximating the flux are implemented in FlowVision. The schemes are based on the reconstruction of variable f in a cel from its average values in the given and surrounding cells followed by translation of the reconstructed values along streamlines. Only general principles underlying the approximation of the convective term are discussed in this section. More details on this subject can be found in se in volume Theory. Figures 12, 13 below demonstrates different ways of reconstructing variable f in a cell. Three-dimensional reconstruction is defined as a linear combination of three one-dimensional functions $f\,k(x_k)$ reconstructed [5]

$$f_i(x) = \sum_{k=1}^{s} f^k(x_k) - 2f. \qquad (7)$$

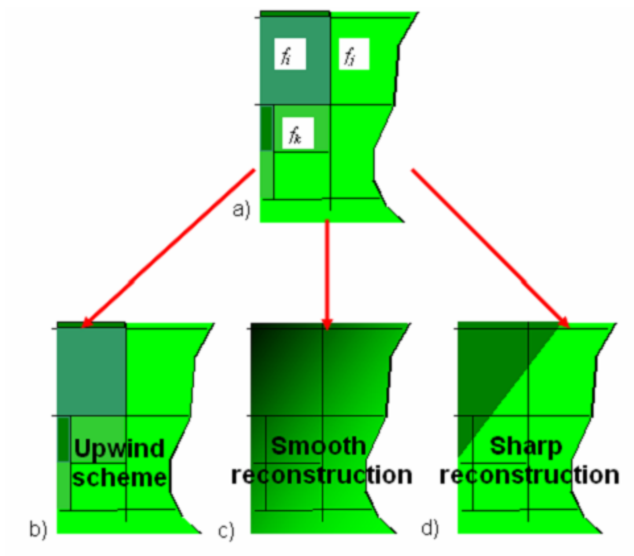


Fig. 12. Methods of reconstructing variable *f* inside a cell: a is the cell-average values; b is the firs-order reconstruction Upwind scheme; c is the high-order Smooth reconstruction; d is the high-order stepwise Sharp reconstruction

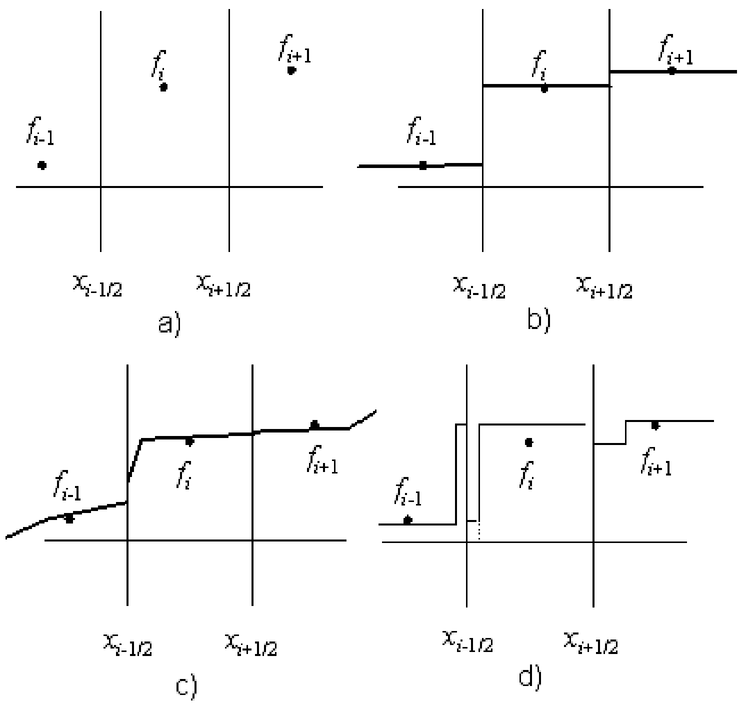


Fig. 13. One-dimensional reconstruction of variable *f* in the cell with boundaries at $x_{i-1/2}$ and $x_{i+1/2}$: a is the volume-average values in cells; b is the firs-order reconstruction Upwind scheme; c is the high-order Smooth reconstruction; d is the high-order stepwise Sharp reconstruction

You can choose an appropriate reconstruction method in the Properties window of the pre-processor element Method Parameters. The most robust method Upwind scheme is set by default (Fig. 14) [4].

The accuracy of the numerical solution of a convection equation strongly depends on the orientation of the flow to the computational mesh. The strongest numerical distortions occur when the flow has diagonal or “skewed” orientation to the cells. To increase the accuracy, a “skewed scheme” is implemented in FlowVision. However, its use augments the time of computing convective terms by ~50 %. The scheme is recommended for swirled flows. To specify the “skewed” scheme for approximation of the convective terms in al the convection and convection-diffusion equations, select check box Skew scheme on tab Steps in the Properties window of the pre-processor element Global Parameters – se figure below. Note that once specified, the scheme will be used in al of the computed domains of the variant (Fig. 15) [6].

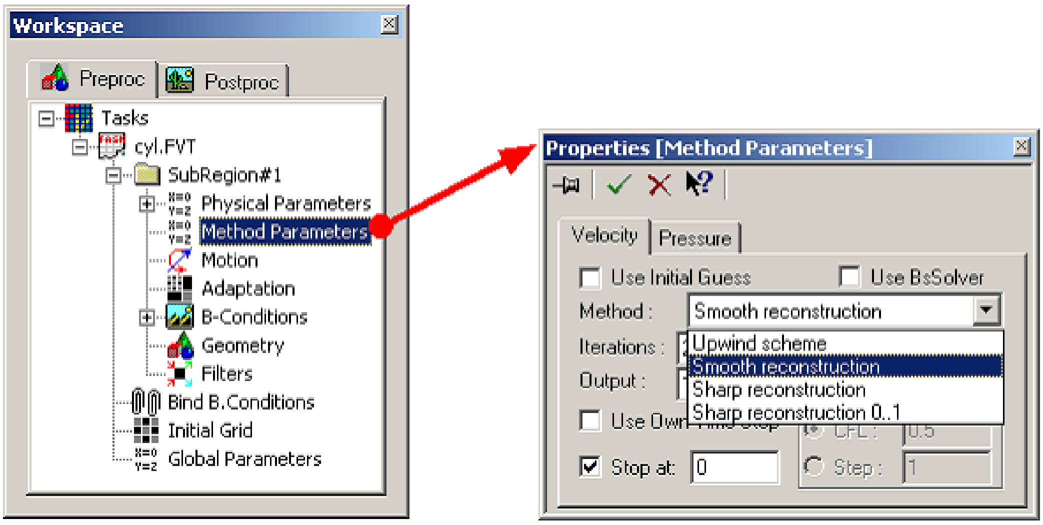


Fig. 14. Choice of numerical scheme for convective term in convection-diffusion equation

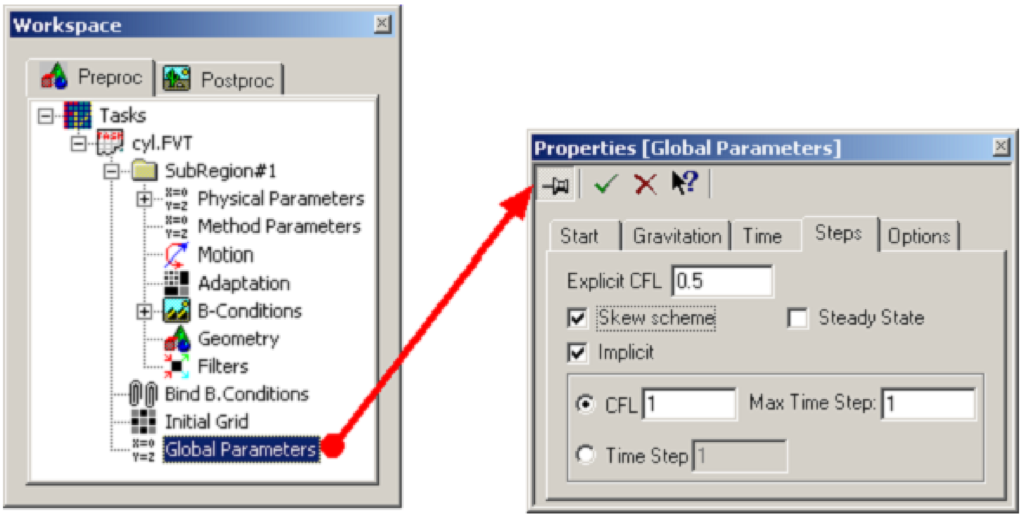


Fig. 15. Scheme wil be used in al of the computed domains of the variant

Step 6. Visualization of results

To visualize the results, click the post-processor tab in the working window. You se several folders: Folder Sights contains 6 elements corresponding to the standard views of the geometry in $\pm X$, $\pm Y$, and $\pm Z$ directions (Fig. 16).

Click the elements one by one and observe the result. Folder Objects contains the geometrical objects used for visualization of the computed variables. Folder Variables contains the independent and dependent computed variables collected from al of the computational domains. Folder Layers contains the graphical objects displaying the variable values in the graphics area. Layers Coordinate System and Solids are created automatically together with the variant. They represent he coordinate system and geometry respectively. To visualize a variable, you have to create an appropriate layer.

For our example we Darrieus rotor received speed data and rotor blades, respectively makes it possible to calculate the power of a rotor, respectively, with its dimensions. In this case, the rotor height of 4 m was obtained in the rotation speed of 34.5 rotations per minute [4].

III. Results of Calculation

The original model of the rotor was shestilopastny two-tier Darrieus rotor height of 4 m and a height of each blade in 1.65 m. Wing profile standard – NACA-016 [1].

At normal pressure and wind speed of 4 m/s (average wind speed in Ukraine) design will make 34.5 rotations around its axis. The angle at which wind interacts with the front plane of the construction is 90° [1].

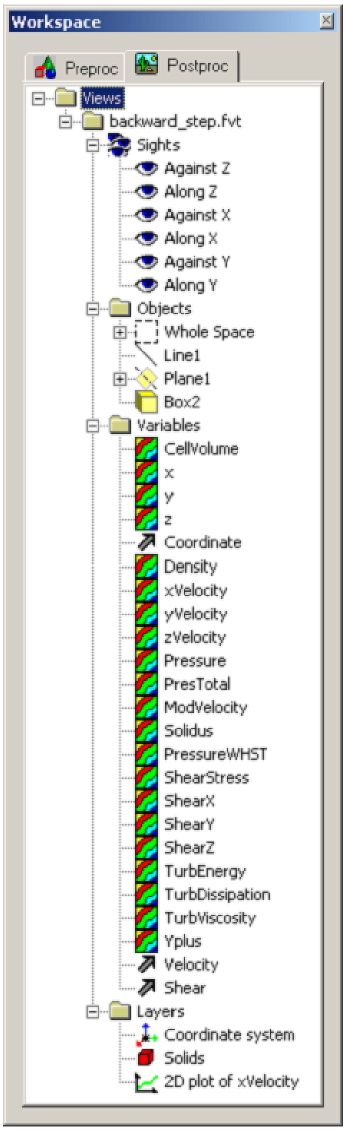


Fig. 16. Results of calculation

When changing the angle to 180° the rotor provides the same amount of rotation, which proves the ability of Darrieus rotor to operate at any wind direction. Increasing wind speed to 5 m/s increases the number of rotations to 39 m/s. From this we can conclude that the increasing the wind speed increases the number of rotations of the rotor, but the dependence is not directly proportional, so increasing the wind speed on 20 % does not lead to 20 % increasing of the number of rotations.

For the simulation of snow or heavy rain when setting initial parameters it is necessary to increase pressure and make it larger than normal atmospheric pressure. But it should be taken into account that the calculation of the parameters does not calculate structural stability to external factors. Graphics for illustration purposes of the results of research (Fig. 17) [3].

On the graph, we can see not quite linear dependence of the speed power to the wind speed (Darrieus rotor) (Fig. 18).

The maximum fence coefficient of power is achieved at speed 5 m/s [3].

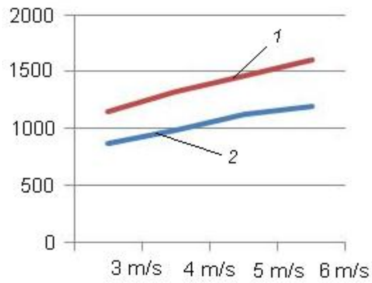


Fig. 17. Power under normal conditions: *1* is a Darrieus rotor; *2* is a combined plant

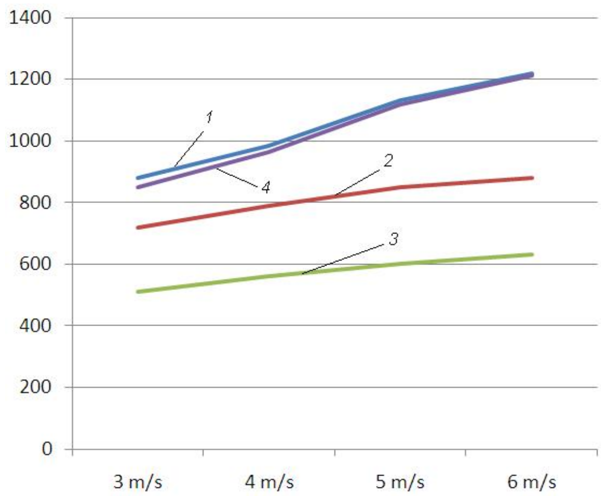


Fig. 18. Dependence of the speed power to the wind speed: *1* is power under normal conditions; *2* is power under the influence of snow; *3* is power under the influence of heavy snow; *4* is power under the influence of rain

For combined power plant characteristic looks like (Fig. 19):

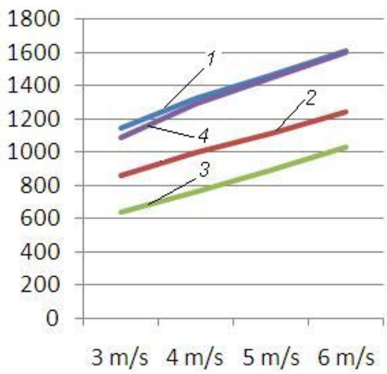


Fig. 19. Characteristic of combined power plant: is power under normal conditions; *2* is power under the influence of snow; *3* is power under the influence of heavy snow; *4* is power under the influence of rain

On the graph we see the influence of power because of the increasing of pressure on the rotor (in this case, the pressure of snow) (Fig. 20). We can see that the received power is much lower. Besides, increasing in power with increasing wind speed is significantly lower in percentage [6].

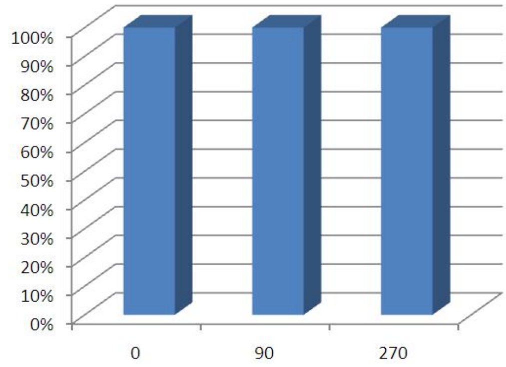


Fig. 20. Influence of power because of the increasing of pressure on the rotor: ■ is dependence on the angle of the wind

This diagram shows the dependence of the amount of received power regardless of the angle of the wind, which interacts with the construction (Fig. 21).

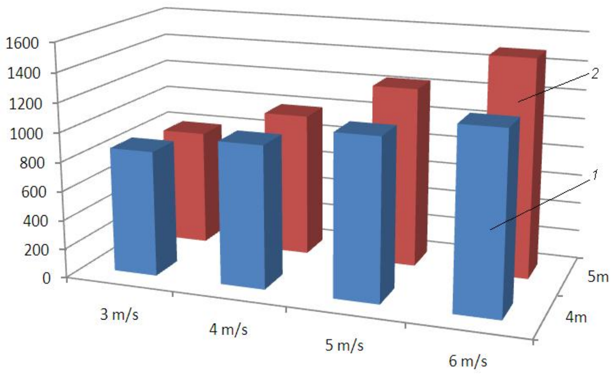


Fig. 21. Dependence of the amount of received power regardless of the angle of the wind: *1* for a 4 m; *2* for a 5 m

The graph shows the dependence of the height of the rotor. If the height of the rotor is 4 m we obtain more power when the wind speed is 3 m/s, than when the height of the rotor is 5 m. This is due to a much greater starting torque of the larger rotor. But with increasing of the wind speed the power of larger rotor begins to exceed significantly the capacity of the smaller rotor [4].

IV. Checks of the Result

To check the results received in flow vision according to this algorithm it has been tested models of other types of rotors, and the result was compared with the readings in manufacturer's documentation. Such models of rotors were tested [7], [8].

Classic horizontal rotor shows the deviations from the index documentation less than 4 % when the wind speed is 3, 4 and 5 m/s. And one of the variations of Darrieus rotor also shows the deviation not greater than 6 % at the aforementioned speed (Figs 22 and 23).

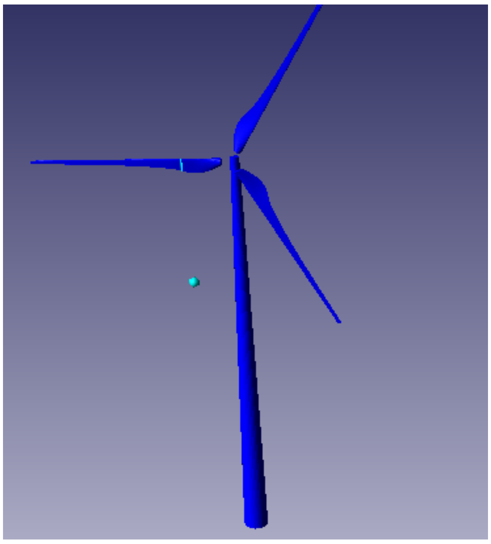


Fig. 22. Classic horizontal rotor

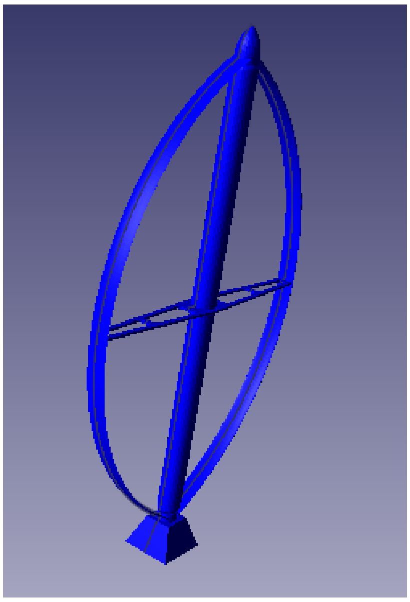


Fig. 23. The variations of Darrieus rotor

Conclusions

This article has Examples of using the software complex flow vision for simulation of rotors without creating a reduced model of the rotor. This allows to carry out the necessary calculations at the minimum of time and money. There were given the Examples of the rotor simulation in different weather conditions and at different wind speeds. To check the accuracy of the results, it was carried out the comparing of the results with the existing rotors documentation [5].

REFERENCES

[1] Kryvtsov, V. S.; Olejnikov, A. M.; Yakovlev, A. I. Inexhaustible energy. Proc. 4 book. vol. 1. Wind power generators. Kharkiv, National Aerospace University, HAI. Sevastopol, Sevastopol National Technical University. 2003. 400 p. (in Russian).

[2] Flow Vision documentation http://flowvision.ru/index.php/public-downloads

[3] Solid Works documentation http://www.solidworks.com/sw/resources.htm

[4] Madsen, David A. (2012). Engineering Drawing & Design. Clifton Park, NY: Delmar.

[5] R&G Composite materials handbook, R&G Faserverbundwerkstoffe GmbH, 2010. 228 p.

[6] Yakovlev, A. I.; Masi, R.; Boyarkin, A. A. “Automatic control of excitation inductor generator for low-speed wind power plant.” *Aerospace Engineering and Technology.* Kharkiv, HAI. 1998. no. 3. pp. 237–241. (in Russian).

[7] Sineglazov, V. M.; Kulbaka A. V.; Boiko, V. M.; “Computer-aided design system of combined wind-power plant” *Electronics and Control Systems.* Kyiv, no. 3(37). 2013. pp. 84–88.

[8] [7] Sineglazov, V. M.; Kulbaka A. V.; Boiko, V. M.; “Computer-aided design system of wind-power plant” *Electronics and Control Systems.* Kyiv, no. 4(38). 2013. pp. 73–78.

Received 15 August 2014.

Sineglazov Viktor. Doctor of Engineering. Professor.
Aviation Computer-Integrated Complexes Department, National Aviation University, Kyiv, Ukraine
Education: Kyiv Polytechnic Institute, Kyiv, Ukraine (1973).
Research area: Air Navigation, Air Traffic Control, Identification of Complex Systems, Wind/Solar power plant.
Publications: more than 450 papers.
E-mail: svm@nau.edu.ua

Boyko Vitaliy. Student.
Aviation Computer-Integrated Complexes Department, National Aviation University, Kyiv, Ukraine.
Research area: wind power plant.
Publications: 2.
E-mail: detroit324@ukr.net

Kulbaka Anton. Post-graduate student.
Aviation Computer-Integrated Complexes Department, National Aviation University, Kyiv, Ukraine.
Education: National Technical University of Ukraine “Kyiv Polytechnic Institute”, Kyiv, Ukraine (2011).
Research area: wind power plant.
Publications: 8.
E-mail: antonkul@ukr.net

В. М. Синеглазов, В. М. Бойко, А. В. Кульбака. Інтегрована система автоматизованого проектування вітро-енергетичної установки

Запропоновано вітро-енергетичну установку, яка представляє комбінацію двох роторів: Дар’є і Савоніуса. Показано, що оптимальна конструкція такої установки можлива тільки у разі використання системи автоматизованого проектування. Розроблено структурну схему системи автоматизованого проектування. Ротор протестований у віртуальній аеродинамічній трубі.

Ключові слова: автоматизоване проектування; вітро-енергетична установка; структурна схема.

Синєглазов Віктор Михайлович. Доктор технічних наук. Професор.
Кафедра авіаційних комп’ютерно-інтегрованих комплексів, Національний авіаційний університет, Київ, Україна.
Освіта: Київський політехнічний інститут, Київ, Україна (1973).
Напрям наукової діяльності: аеронавігація, управління повітряним рухом, ідентифікація складних систем, вітроенергетичні установки.
Кількість публікацій: більше 450 наукових робіт.
E-mail: svm@nau.edu.ua

Бойко Віталій Миколайович. Студент.
Кафедра авіаційних комп’ютерно-інтегрованих комплексів, Національний авіаційний університет, Київ, Україна.
Напрям наукової діяльності: вітроенергетичні установки.

Кількість публікацій: 2.
E-mail: detroit324@ukr.net

Кульбака Антон Валерійович. Аспірант.
Кафедра авіаційних комп’ютерно-інтегрованих комплексів, Національний авіаційний університет, Київ, Україна.
Освіта: Національний авіаційний університет, Київ, Україна (2011).
Напрям наукової діяльності: вітроенергетичні установки.
Кількість публікацій: 8.
E-mail: antonkul@ukr.net

В. М. Синеглазов, В. Н. Бойко, А. В. Кульбака. Интегрированная система автоматизированного проектирования ветро-энергетической установки
Предложена ветро-энергетическая установка, которая представляет комбинацию двух роторов: Дарье и Савониуса. Показано, что оптимальная конструкция такой установки возможна только при использовании системы автоматизированного проектирования. Разработана структурная схема системы автоматизированного проектирования. Ротор протестирован в виртуальной аэродинамической трубе.
Ключевые слова: автоматизированное проектирование; ветро-энергетическая установка; структурная схема.

Синеглазов Виктор Михайлович. Доктор технических наук. Профессор.
Кафедра авиационных компьютерно-интегрированных комплексов, Национальный авиационный университет, Киев, Украина.
Образование: Киевский политехнический институт, Киев, Украина (1973).
Направление научной деятельности: аэронавигация, управление воздушным движением, идентификация сложных систем, ветроэнергетические установки.
Количество публикаций: более 450 научных работ.
E-mail: svm@nau.edu.ua

Бойко Виталий Николаевич. Студент.
Кафедра авиационных компьютерно-интегрированных комплексов, Национальный авиационный университет, Киев, Украина.
Направление научной деятельности: ветроэнергетические установки.
Количество публикаций: 2.
E-mail: detroit324@ukr.net

Кульбака Антон Валерьевич. Аспирант.
Кафедра авиационных компьютерно-интегрированных комплексов, Национальный авиационный университет, Киев, Украина.
Образование: Национальный авиационный университет, Киев, Украина (2011).
Направление научной деятельности: ветроэнергетические установки.
Количество публикаций: 8.
E-mail: antonkul@ukr.net