Machinery & Energetics
Vol. 13, No. 3. 2022

Journal homepage:
https://technicalscience.com.ua/en

UDC 631.331
DOI: 10.31548/machenergy.13(3).2022.62-72

Victor Melnik*, **Alexei Zelensky, Andrew Zelensky**

State Biotechnological University
61002, 44 Alchevskikh Str., Kharkiv, Ukraine

Numerical Simulation of Gas-Dynamic Processes in the Centrifugal Radial Fan of Seeding Machines

Abstract. Increasing the efficiency of crop production, where a significant place is occupied by row crops (such as corn, sunflower, soy, sorghum, sugar beet, etc.), is inextricably linked with the introduction of energy- and resource-saving technologies to ensure high-quality and timely implementation of production processes. Sowing is an important technological process. To increase the speed, as a consequence, productivity and, most importantly, the quality of field work, precision seed drills with pneumatic systems with improved characteristics of vacuum generators are currently used. At the same time, the vacuum systems of such seeding machines are most often based on the use of a radial centrifugal fan. Therefore, an important task is to design the fan, considering the harmonisation of its parameters with the parameters of the seed drill in general, which requires a large number of tests and inspections. The use of full-scale models (test tables) requires significant time and resource costs. Based on numerical modelling of gas-dynamic processes, it becomes possible to significantly reduce the number of field experiments, and rather determine the rational form of a centrifugal radial fan. The purpose of this study is to consider the method of applying numerical modelling of gas-dynamic processes in a centrifugal radial fan using computer-aided engineering (CAE) packages. Application of computational fluid dynamics (CFD) methods for discretisation of solutions of differential equations taking into account Euler equations and Navier-Stokes equations. Numerical turbulence modelling based on k-ε models, k-ω models and Shear Stress Transport (SST) models. The results of calculating the three-dimensional field of velocities and pressures in the working area of the fan are presented, rational geometric characteristics and aerodynamic characteristics (dependence of the differential pressure on air flow) are determined. This approach to designing a centrifugal fan allows significantly harmonising its parameters at the development stage and reducing the time for implementing new projects

Keywords: numerical modelling, k-ε model, k-ω model, SST model, finite element method (FEM), finite volume method (FVM), CAD editor

Article's History: Received: 23.02.2022; Revised: 30.06.2022; Accepted: 11.08.2022

INTRODUCTION

Nowadays, high-pressure centrifugal radial fans (CRF) are widely used in agricultural machinery. They are one of the most important components of the pneumatic system of modern precision seeders. Often CRFs operate in a wide range of air flow values, which affects their productivity and, accordingly, the stability of pneumatic seed drill systems in general. In the end, all this affects the quality of work and productivity of seeding machines and, consequently, the speed and quality of sowing operations.

The use of innovative design solutions on seed drills allows achieving stable sowing quality, which leads to the establishment of the most favourable conditions for sown seeds and, as a result, to uniform germination. The latter, in turn, provides an increase in yield while minimising costs. The modern pneumatic seed drill system meets a wide range of requirements and, in particular, allows working in a precision farming system. Modern precision seed drills provide the possibility of flexible section-by-section adjustment of all seeding processes, in particular, they allow fine-tuning and accurately maintaining the seeding rate. The creation of a reliable and efficient CRF, as the basis of the pneumatic system of the seed drill, will further improve the quality indicators of its operation. This requires a detailed investigation and a deep understanding of the workflow that occurs between the interblade channels of the CRF. Applying the

Suggested Citation:
Melnik, V., Zelensky, A., & Zelensky, A. (2022). Numerical simulation of gas-dynamic processes in the centrifugal radial fan of seeding machines. *Machinery & Energetics,* 13(3), 62-72.

*Corresponding author

acquired knowledge in the design of CRF will allow avoiding mistakes, which will lead to an increase in gas-dynamic stability, performance facilitation, and overall efficiency of the developed fans.

As a result of the development of new promising pneumatic systems, the main source of obtaining the characteristics of a new fan was previously an experiment, which was unnecessarily expensive. A full-scale experiment requires the production of a large number of experimental parts of various, sometimes very complex geometries, and this, in turn, involves the use of high-tech equipment with a significant amount of time. At the same time, it is not possible to obtain comprehensive information about the flow parameters at all points of the studied region of the interblade channel [1-3]. All this affects the quality of the design and the cost of finishing the CRF.

Due to the growing progress in computer technology, flow studies using computational gas dynamics methods have become more promising methods for calculating the processes occurring in the flow part of the fan. Most often, such methods are based on the numerical solution of systems of Navier-Stokes equations [4].

The main advantages of numerical methods are:

– low cost (no need to create a test table for conducting experiments, which means that the need for appropriate special equipment, personnel training, etc. is eliminated);

– high information content of the simulated process, obtaining comprehensive information about all parameters at all points of the analysed flow area;

– safety of research processes;

– reducing the number of experiments to finalise CRF parts, which has a positive effect on the time and cost of developing the final product.

Considering the CRF as an aggregate in the pneumatic system of the seed drill, then the proposed methodology is presented for numerical modelling of gas-dynamic processes occurring in the interblade channels of the fan, and the determination of its optimal design and characteristics.

The process of development of seed drills shows the change of seeding machines that operate on a mechanical principle with the help of gravity forces to machines that use air flow. Long-term use of machines that use air flow during seeding [5] show an increase in the quality of seeding, an increase in the yield of agricultural crops with a decrease in technological and operating costs during their production. The quality of sowing seeds of row and grain crops can be implemented by improving the quality of the pneumatic system. These topics were considered by V.P. Chebotarev [6], V.V. Alt [7], S.G. Yasir [8] et al. The pneumatic system of the precision seed drill consists of a pneumatic seed drill, pneumatic pipes, and a fan. One of the main tasks is to ensure the quality of operation of the pneumatic system, in most cases, the vacuum system, and, in particular, to increase the stability of the CRF.

To solve this problem, much attention is usually paid to the development of an energy-efficient design and workflow of the pneumatic system. But at the same time, systematic studies show that the methodology for designing the CRF as an aggregate in the context of a pneumatic system of seeding machines has not been sufficiently developed.

The purpose of the study is: to consider the application of the sequential design scheme of CRF; to consider the numerical investigation of fan models using modern computer-aided engineering (CAE) software suites using computational fluid dynamics (CFD) methods [9-11]; to create a solid-state and mesh model; application of the method for describing gas dynamic processes, and a method for solving problems of gas dynamics based on discretisation methods for solutions of Euler differential equations and Navier-Stokes equations [12].

MATERIALS AND METHODS

A typical process of CRF modelling sequence is shown in Figure 1 [9]. To achieve the goal of developing a new CRF model, it is necessary to complete the following steps:

– analysis of existing models of fans of pneumatic precision seeders;

– creation of a technical task for model development;

– mathematical statement of the design problem;

– determination of geometric parameters, creation of a geometric scheme and profile of the fan impeller;

– implementation of 3D modelling of the fan model under study;

– substantiation of the choice of a mathematical model describing air flows and the solution method;

– solving the equations of the selected mathematical model of air flow;

– analysis of the results obtained and optimisation of geometric shapes of the flow part of the projected CRF.

The study considers the processes occurring in the flow part of the CRF. The fan blades, which form a circular mesh, rotate the air located in its channels [1; 13]. Under the influence of the resulting centrifugal forces, the air becomes accelerated, its density, pressure, and temperature increase. In the diffuser, the air velocity decreases, and pressure and temperature increase due to a decrease in kinetic energy. Gas flow currents can acquire such modes as: laminar, turbulent, and mixed laminar-turbulent.

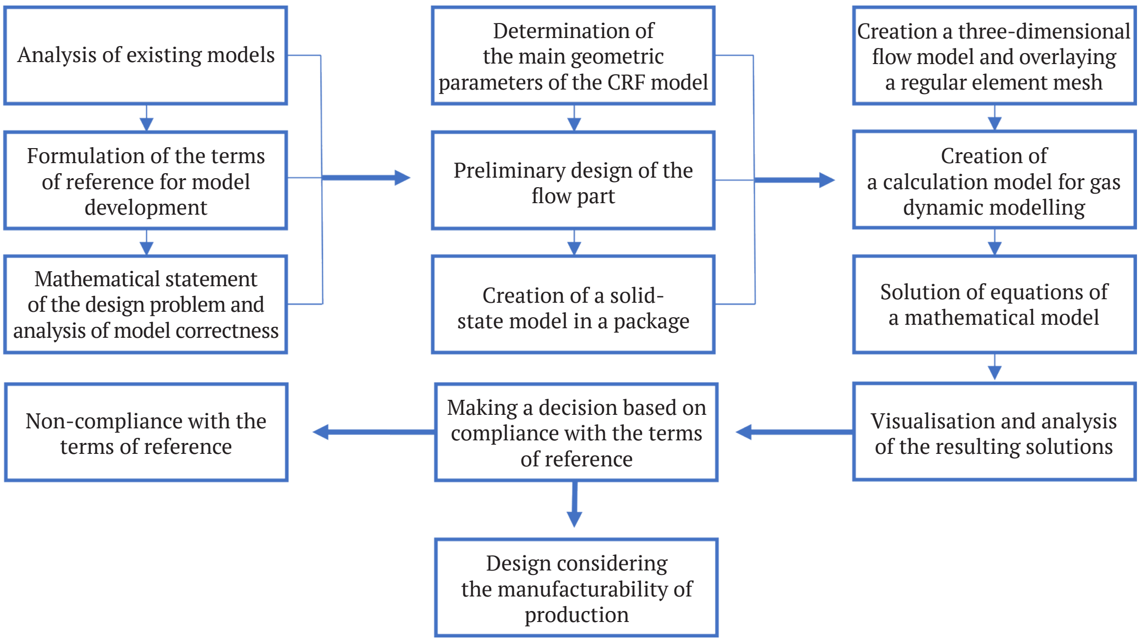


Figure 1. Modelling sequence of the process occurring in a centrifugal radial fan

The main difficulty in describing and calculating the air flow dynamics is to describe the turbulence of the flow. Turbulence is a three-dimensional non-stationary motion in which, due to the stretching of vortices, a continuous distribution of velocity ripples is created in the wavelength range from the minimum, determined by viscous forces, to the maximum, determined by the boundary flow conditions [4]. Under turbulence conditions, due to the intense mixing of air flows between different regions, the flow develops faster than laminar flow, the exchange of mass, momentum, and energy. In turn, part of the kinetic energy of the flow is converted into the kinetic energy of turbulent ripples, which then turns into heat, and these processes are irreversible [14]. Turbulence occurs in a continuous medium where small spatial fluctuations exceed the free path length of molecules.

The main difficulties in calculating the viscous incompressible flow are associated with a wide range of changes in the turbulence scale. When directly calculating the Navier-Stokes equations for a three-dimensional turbulent flow, large computational resources are required, and when calculating viscous flows, it becomes necessary to use a fine mesh with subsequent time step grinding, which also entails an increase in the computational resources used. Therefore, the main task is to select a method for solving gas dynamics equations that will reduce the necessary computational resources without losing accuracy in calculations.

Next, the study considers the process of modelling air flows on the CRF (Fig. 2). After creating a model of the processes occurring in the CRF and in the pipelines of the pneumatic system, a mesh approximation of the design area is performed. It is possible to create both an unstructured calculation mesh and a block-structured one. Then the model of a continuous medium is selected (in this case, air). The environment can be compressed or uncompressed. The boundary conditions are substantiated, in this case for moving and stationary elements, and also the wall flows of the model medium flow are described (the process of turbulence occurrence). During the execution of decisions, the accuracy is evaluated. At the same time, mathematical and physical estimates are made, the error during modelling is estimated, and the iterative solution is discretised. The results obtained for CRF are analysed. Such analysis can be graphical or numerical. The analysis is performed using the method of statistical analysis, correlation, variance, or factor. Based on them, conclusions can be drawn about further actions when designing the model: send it to production or refine it. Considering the gas flows that occur in a pneumatic system, there are two main approaches to modelling:

– direct numerical simulation of turbulent flows (DNS) consists in solving complete Navier-Stokes equations [15];

– modelling of flow using the Navier-Stokes mean equations over time (RANS), space (LES), and hybrid approaches (DES).

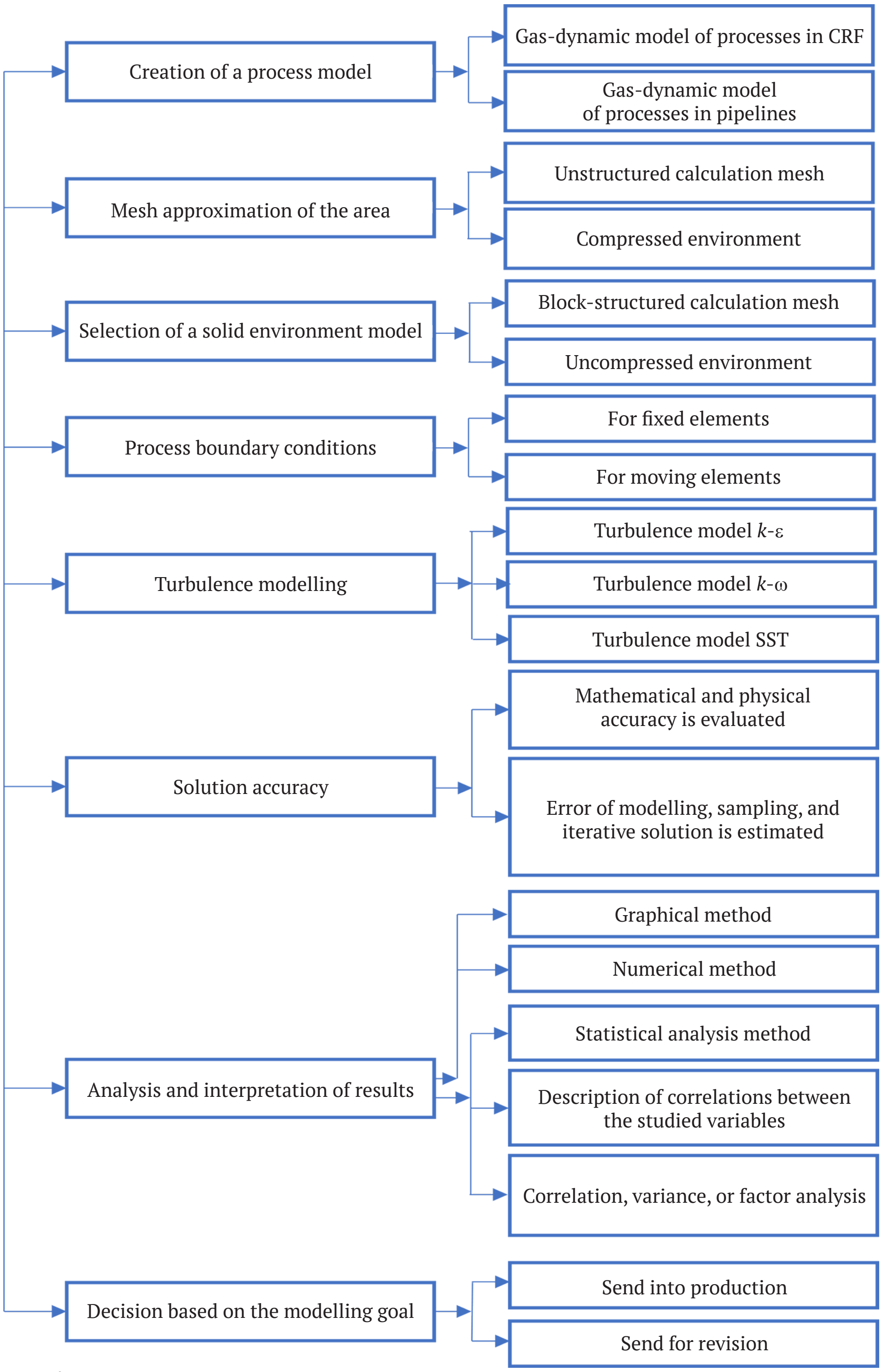


Figure 2. Modelling sequence of the process occurring in a centrifugal radial fan

Next, the study considers the pneumatic system of a precision seed drill (Fig. 3), consisting of CRF 1, a tree of pneumatic channels 2, and a group of seeding machines 3. Air enters CRF 1 through the inlet device 4, and goes outside through the outlet section 5 of the vacuum created by fan 1. Ambient air enters the channels 2 through the seeding machines 3, and then, in the form of a single flow, through the input device 4 it moves to the rotary part of the CRF 1 and then through the output section 5 (scroll housing) it is brought out again.

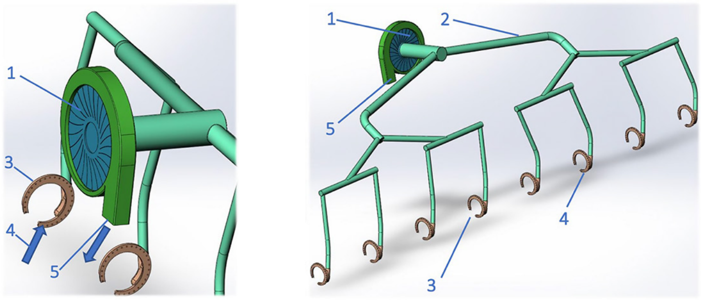


Figure 3. Design diagram of the pneumatic system of a precision seed drill with a centrifugal radial fan: 1 – fan; 2 – tree of pneumatic channels; 3 – seeding machines (air intake area); 4 and 5 – inlet device and outlet section of the fan

In the input device of the CRF, a laminar-turbulent transition is observed in wall currents. The boundary layer on the surface is a thin layer with a fast variable rate from zero to 99.5% of the incoming flow rate. In the boundary layer, due to the so-called Tollmin-Schlichting instability [4], transverse vortices are formed which undergo secondary instability and form Lambda structures downstream. In the future, they interact with each other and experience tertiary instability. Eventually, this leads to a loss of regular shape, and they become stochastic [4; 16; 17]. Another important feature of a turbulent boundary layer is the presence of a low-velocity jet stream that occurs due to the induction of Lambda structures. To numerically solve partial differential equations, the study used the finite element method (FEM). According to this method, the calculated area is divided into a set of finite sub-regions (elements) using a mesh. For each of these elements, the type of approximating function is reasonably chosen. In this case, outside of its element, the approximating function is equated to zero. The value of the function at the boundaries of elements (nodes) is obtained as a solution to the problem.

At the initial stage, when modelling the process occurring in the flow part of the projected CRF, such values are unknown in advance. Then, based on the tools of geometric design of the CAD package, a 2D model is formed, and then a 3D model (solid-state model) of the CRF (Fig. 4). Then, based on a three-dimensional solid-state model, using the operations of subtracting the volumes of the flow part and reducing insignificant parts and elements of the object under study, the area of air flow in the calculated CRF channel is distinguished (Fig. 5). In this case, the model under study is divided into several volumes (static and rotating sections of the calculated area). They will later be set by different boundary conditions. In this case: position 1 – the first input section, the volume that does not rotate; position 2 – the second section, the rotary part, the volume that rotates; position 3 – the third output section, the volume that does not rotate.

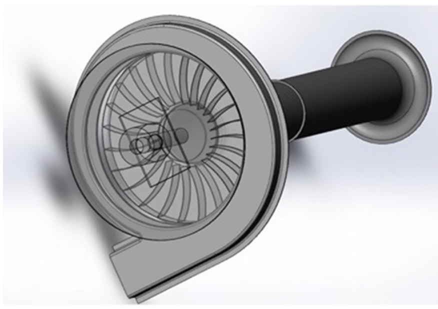


Figure 4. Solid-state model of centrifugal radial fan

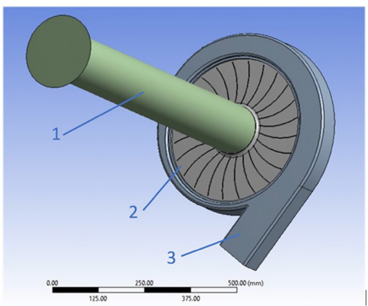


Figure 5. Air flow area of a centrifugal radial fan in the design channel: 1 –the inlet section is stationary (the first volume); 2 – the movable part (the second volume); 3 – the outlet area is stationary (the third volume)

Modelling a finite element mesh it is one of the most important stages of the computer engineering modelling process. It is used to divide the geometric area of calculation of the studied flow into elements, control volumes, etc., and also defines discrete points that correspond to the solution of partial differential equations. CAE packages offer two types of end-element mesh usage: structured and unstructured.

Structured elements are hexahedral elements, and unstructured elements are hexahedral, tetrahedral, or polyhedral elements. When dividing the study area into a calculated mesh in the regions of large velocity, pressure, and temperature gradients, a high-density mesh is used. In turn, the faces of the final volumes should be oriented by the flow lines of the flow under study.

In the sampling process, an approximation should be performed for the derived values at the nodal points (centres) of the control volumes of the resulting finite element mesh. This is because when considering internal flows for which the velocity components at the boundary are known and the pressure is unknown, it is easier to use unknowns at the centres of mass of elements. In this case, the number of points for calculating the pressure coincides with the number of points for calculating the velocity components. In addition, in the control volume centres, the values of the unknown function itself are not held, but only mediate the value of the control volume [3; 18; 19]. In turn: 1) the numerical solution obtained based on splitting the calculated space into discrete elements is approximate; 2) to improve the accuracy of the solution, it is broken down into smaller elements; 3 there is an accumulation of error; 4) the splitting method should ensure stability or convergence of the solution to the exact value when the splitting step decreases; 5) it is proposed to use the Neumann splitting method [4; 5; 20].

Next, for the three volumes under consideration, a mesh is created using the CAE package mesh generator. An automated algorithm is used to build a high-quality hexagonal mesh. After setting the parameters and requirements for the mesh, considering the requirements for probabilistic areas of turbulence, a three-dimensional finite element mesh is constructed: depending on the radius of curvature of the body; considering the number of given cells of elements, considering the thickening of the number of layers along the boundary layer and the size of cells, etc.

In turn, the current in the channels of the flow part of the model under study is conventionally divided into the region of the turbulent flow core and the region of the boundary layer. It significantly manifests friction forces, which is why it is characterised by a significant velocity gradient (the velocity varies from zero on the surface of the body's flow to the flow rate in the flow core) [3]. The thickness of the boundary layer usually corresponds to the distance from the wall, at which the flow rate will differ from the external flow rate by 1%.

In addition, the quality of the mesh affects the accuracy, convergence, and speed of obtaining a solution. The created mesh is pre-calculated, namely the number of elements and nodes. If necessary, considering the method of wall functions, a layer-by-layer reconstruction with an increase in the density of critical sections of the model. To investigate the operating modes of the CRF that significantly differ from the nominal (cruising) ones, the requirements for the quality of mesh breakdown are increased.

Having finally decided on the mesh parameters, a mesh model was obtained, which is shown in (Fig. 6).

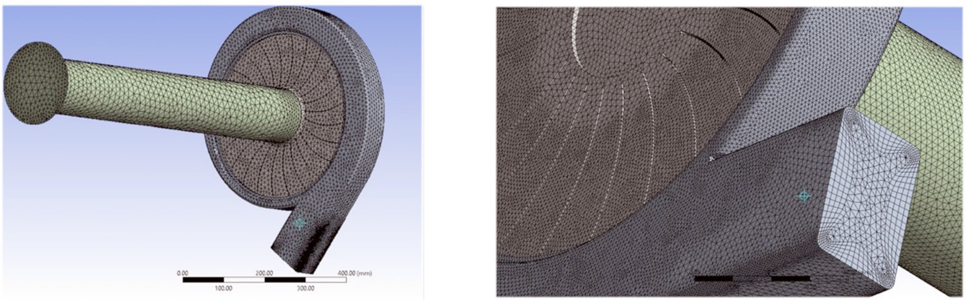


Figure 6. Mesh model of the flow part of a centrifugal radial fan

Setting calculation conditions. The calculation conditions, namely, the boundary conditions for the calculated meshes of elements of the flow part of the CRF model under study, in which it is necessary to calculate the air flow, are introduced into the processor of the PRE CAE package. At the same time, the calculated model under study consists of three areas [8; 14; 15].

Considering the fan model under study, it is assumed that, from a mathematical standpoint, the flow of the fan blade with air flow determines the system of Navier-Stokes equations [19]. It is recorded for the volume of air in question, which envelops the scapula after removing the volume of the scapula itself from it. The solution of this system gives the desired parameters of air flow in the area of interest.

During the simulation, 2 parametric turbulence models are used – k-ε (k – kinetic energy of turbulence, ε – dissipation) and SST (Shear Stress Transport). The latter is a combination of the k-ε and k-ω models of turbulence (ω –specific dissipation of turbulence), where the equation k-ε is used to calculate the flow in the free flow, and the equation k-ω of the model is used in the region near the walls [20]. The Reynolds Navier-Stokes equation (RANS) [12] are used to model turbulence with the Boussinesk approximation. The Businesk approximation (with the introduction of turbulent viscosity) consists of using Reynolds averaging of the Navier-Stokes equations. In the latter, the instantaneous velocity values are represented as the sum of the ripples of the mean over time, with the introduction of additional transfer equations for the characteristics of turbulence due to the turbulent kinetic energy *k* of the flow and its dissipation rate ε. In the end, 2 parametric RANS models *k*-ε and SST are chosen with turbulence models that are reliable, economical, and have acceptable accuracy for a wide range of turbulent flows with submodels for flow compressibility. In addition, the software products used implement a wide range of mathematical models that allow fully simulating the physical processes that often take place found in practice.

The following were chosen as the boundary conditions for the model under study: operating flow type – ideal gas; flow mode – steady state; flow without heat transfer; reference pressure 101,325 Pa; turbulence model *k*-ε or SST; rotor speed – 5,000 rpm.

As the boundary conditions at the inlet and outlet of the flow part: "Outlet pressure and mass air flow": outlet pressure (overpressure) – Free, i.e., 0 pa; air flow rate (Mass Flow Rate) – 0.494 KG/s [9]; Total Temperature – 288.15 K; combination of areas – select "on surfaces".

For further calculation, the appropriate surfaces were selected, set names for the calculated areas, and workflow parameters for the Inlet, Outlet sections, and the three areas marked: Hub – wall, Shroud – wall, and "Wall" – wall. The rotation was set for conditions of the "Wall" type (Figs. 7, 8).

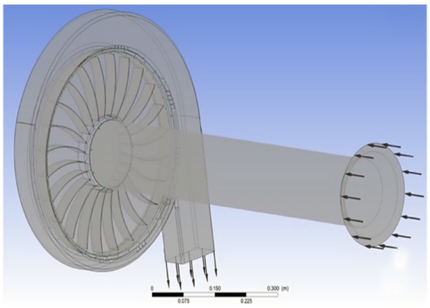


Figure 7. Calculation model with boundary conditions

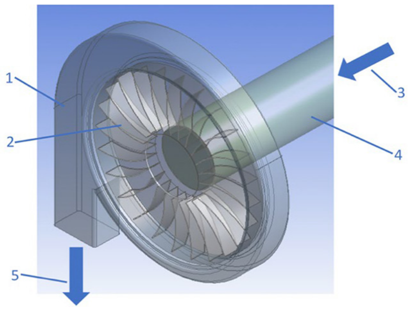


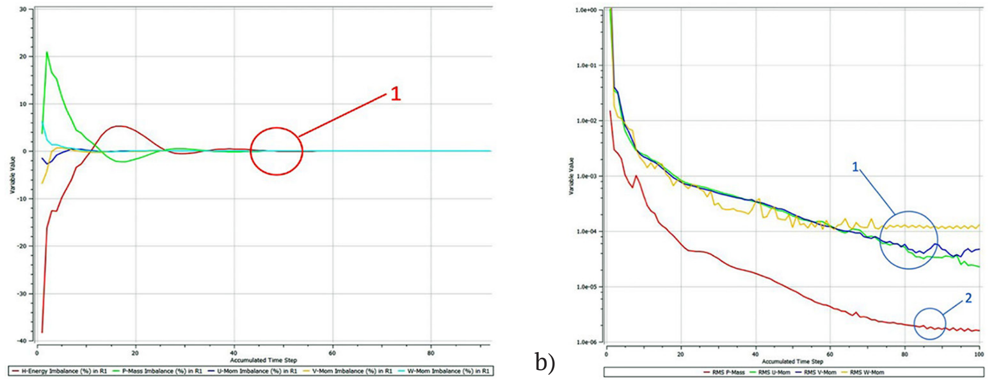
Figure 8. Setting of boundary conditions: 1 – air channel – fixed walls; 2 – impeller – movable walls; 3 – Inlet flow; 4 – air channel – fixed walls; 5 – outlet flow – mass flow rate

In addition, an important factor in gas-dynamic design is the reduction of pressure losses, uniformity of distribution, and the stabilisation of the values of the speed and flow rate of the gas flow in the channel [21-23]. When discretising the equations of a mathematical model, the weighted mismatch method was used. In this method, the solution is considered to converge with very small values of inconsistencies. The convergence criterion for solving a system of Reynolds equations is the value of the velocity values in each equation at the end of each iteration for a value less than RMS -1.0 E-6 (root-mean-square value).

RESULTS AND DISCUSSION

After all the boundary conditions have been set, the calculation requirements are set. Next, the process of numerical solution of the problem is started.

During calculations, a large array of calculated data was obtained (Fig. 9). Using them, the air flow in the flow part of the CRF is simulated. This allows seeing a characteristic picture of changes in gas-dynamic parameters, values of velocity, flow rate, pressure, and density of air flow distribution over the cross-sections of the study area.



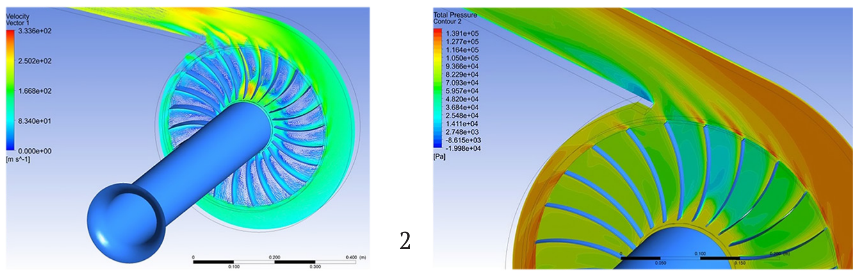
a) b)

Figure 9. Calculation results: a – calculation of a three-dimensional model of the air flow of a centrifugal radial fan (1 – an imbalance of solutions less than 1%); b – the root-mean-square value of the mass flow rate and moments

Since, when solving the equations of conservation of mass, momentum, energy, etc., the final solution must preserve these quantities, it should be borne in mind that for a numerical representation of a physical system, the imbalance of the CFD solution must be quite small, but it should not be zero. It is recommended that the desire for an imbalance in the solution should be less than 1% [24-26] (Fig. 9a). In Fig. 9b sections 1 and 2 of the non-coupling of integral parameters tend to be linearised, values oscillate, and the solution of moving convergence occurs. The solution is considered to converge when the values of the mean-square (RMS) inconsistencies are less than 1.0e-4...-1.0e-5. The convergence of solutions can be influenced by the turbulence model, the quality of the design mesh, wall functions, boundary conditions, air flow mode, etc. [27-29].

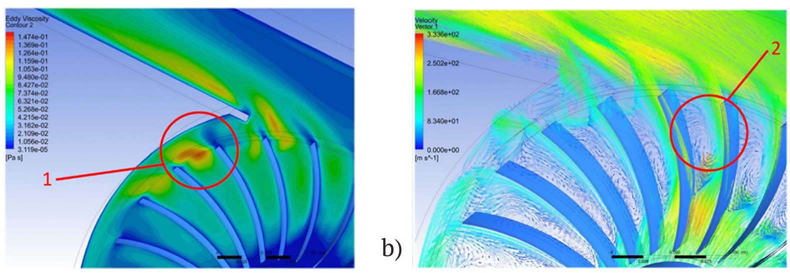
Analysing the obtained data, there is a possibility of determining the zones of disturbances (vortex generation) of the air flow, which negatively affects the nature of the flow in the channel, contributing to losses of velocity, pressure, and air flow (Fig. 10). Similar approaches were used in the design of CRF and analysis by G. Ronald, I.L. Lokshin [30-31]. Vortex generation zones and relative vortex regions were considered, which requires improvement and introduction of structural changes to the surfaces and a deeper study of these regions to improve the conditions of air flow. It is the use of numerical modelling of gas-dynamic processes in a centrifugal radial fan using CAE packages that facilitated a more thorough investigation of the processes that occur in the event of flow disturbances and turbulence.

Based on the obtained calculation results, the processes that take place in the flow part of the CRF are modelled. At the same time, the study relies on the data of changes in pressure, speed, and other parameters of interest along the flow part of the fan (Fig. 10, Fig. 11a) [2].



1 2

Figure 10. Picture of changes in the velocity and pressure vector of a centrifugal radial fan for the *k*-ε turbulence model 1 – separation loss and 2 – edge loss (shown in red circle)



a) b)

Figure 11. Turbulent viscosity (a) and flow velocity vectors (b) in the flow part for a centrifugal radial fan (*k*-ε model) – the red circle shows the regions of relative vortices: 1 – the region of increased turbulent viscosity; 2 – the region of relative vortices

In addition, for example, the fields and velocity vectors along the flow part of the CRF can be considered in detail (Fig. 11b). Relative vortex (Fig. 11b) is formed in a rotating (ideal, according to the mathematical model) relative air flow in the interblade channel. Gas molecules cannot perform rotational movements in the interblade channel when flowing without friction. Therefore, they retain their direction in absolute space and during the rotation of the impeller, where each molecule moves in a circle.

Figure 11 shows that when the flow is closed in the interblade channel, a relative rotational motion is formed with the same angular velocity and the direction of rotation, the opposite direction of rotation of the impeller.

This is a relative vortex of gas molecules orbiting a calm core. Under the influence of an axial vortex, the circumferential component of the output velocity will decrease. Figure 12 shows the possible losses that occur in the fan in a systematic form, partially shown in (Fig. 10, 11).

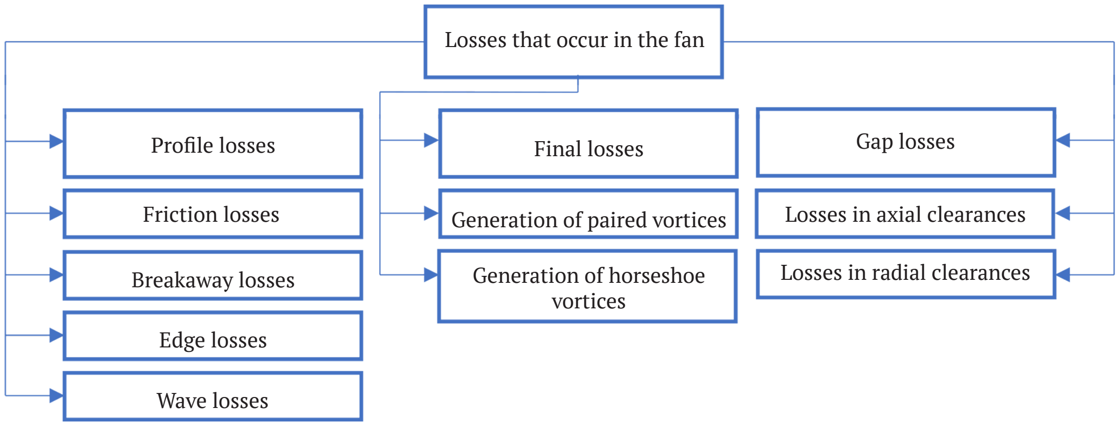


Figure 12. Losses that occur in a centrifugal radial fan

Fan losses can be divided into three groups: profile losses, end losses, and gap losses. Profile losses include losses that occur in a plane lattice: friction losses in the boundary layer, vortex losses when the flow breaks off on the profile, vortex losses behind the initial edge – edge losses; losses occur in compaction jumps – wave losses. Final losses are losses that occur in areas of the interblade channel adjacent to the end surfaces. As a result of flow in the convex surface at the top and bottom of the fan blade, at the junction with the end wall, local thickenings of the boundary layer occur. When interacting with the main flow, the thickened layers of the boundary layer begin to break off and form paired vortices rotating in the opposite direction. In addition, during turbulent separation, the air flow that forms the boundary layer twists and forms a vortex system, in which the ends bend down, taking a horseshoe shape. Losses in clearances include: losses in radial and axial clearances.

The results of the mathematical modelling of various fan models presented in Table 1 are obtained by multiple repetitions of calculations with different specified boundary conditions and subsequent mediation. To present the methodology for modelling the CRF workflow, there were no differences in the geometry and design features of the fans under study. The rotor speed is assumed to be 5,000 rpm, and the turbulence model is *k*-ε. By conducting research on a fan with counterclockwise curved blades model No. 01400Z24R, the lowest air consumption and low efficiency were obtained. When changing the direction of the blades (model No. 02400Z24LS), an increase in air consumption and efficiency was obtained. Improving the blade profile by changing the entry angle by 34° (model No. 03400Z24R50), the highest efficiency and highest air consumption were obtained. In model No. 04400Z24SB with straight blades and an entry angle of 44°, low air consumption and low efficiency were obtained.

Table 1. The result of gas-dynamic modelling at n-5,000 rpm and the turbulence model *k*-ε

Fan model	Air consumption, G_f, kg/s	Degree increased blood pressure, π_f	Efficiency, η
No. 01400Z24R	2.69	1.03	0.27
No. 02400Z24LS	2.81	1.05	0.57
No. 03400Z24R50	2.89	1.13	0.79
No. 04400Z24SB	2.55	1.04	0.423

A graphical interpretation of the relationships between the main parameters of the CRF (fan characteristic) is shown in (Fig. 13). Graphs allow analysing changes in the flow rate (Fig. 13a) and the pressure generated by the fan (Fig. 13b) in the middle section of the fan (Fig. 13: a – shows the change in velocity; b – shows the change in pressure along the cross-section of the interblade channel in the region of relative vortices) [21]. In Figure 13a, sphere 1 shows the place where the flow exits the braking area, and in Figure” 13b, the area between 1 and 2 shows the vortex generation area.

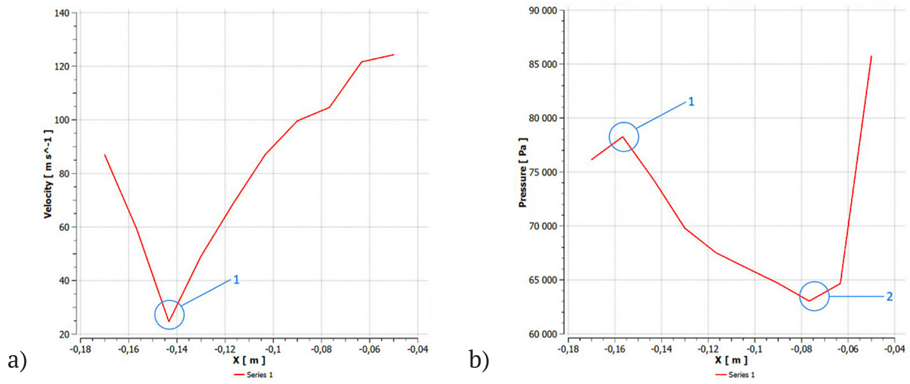


Figure 13. Area of occurrence of relative vortices:
a – distribution of flow velocity along the Section; b – distribution of flow pressure along the section

When considering the field of velocity vectors (Fig. 11b) there is an uneven flow along the height of the region of generation of reverse currents of axial vortices, which contributes to the generation of flow closure sites. All this suggests that it is necessary to deal more carefully with the profiling of the blades and the flow part of the CRF. Finding the optimal approach (changing the shape and size, angles of flow entry and exit in the fan crown) will allow minimising the total loss values.

CONCLUSIONS

Based on the presented methodology for calculating gas-dynamic parameters, it is possible to assess the nature of air flow in the flow part of the CRF, draw conclusions about the shape and size of pipelines, evaluate the design quality of the centrifugal wheel itself, that is, installation angles, dimensions, thickness, a number of blades, noise and safety factors.

Studies show that during design, achieving the desired air flow modes, high flow rates in the interblade channels of the CRF and, ultimately, acceptable values of the final parameters of the CRF can be achieved through careful profiling and the introduction of new design solutions.

The use of CAE software packages based on CFD methods in modelling the gas-dynamic characteristics of gas flow in the flow parts of CRF and pneumatic systems, in general, is very effective. This allows predicting the final parameters and significantly reducing the time and resources spent on fine-tuning such systems. In further research, when designing centrifugal radial fans, it is desirable to analyse the influence of various turbulence models on the quality of calculating air flow flows, adn conduct a comparative analysis with experimental data. This approach should be recommended for widespread use.

REFERENCES

[1] Kalinkevich, N.V., & Gusak, A.G. (2011). *Theory of turbochargers*. Sumy: SSU.
[2] Rathakrishnan, E. (2010). *Applied gas dynamics*. Hoboken: Wiley.
[3] Solodov, V.G. (2017). *Application of the ANSYS software package for solving fluid dynamics problems*. Kharkiv: Publishing house of KHNADU.
[4] Schlichting, H., & Gersten, K. (2017). *Boundary-Layer theory* (9th ed.). Berlin: Springer-Verlag.
[5] Precision seeder for row crops SCH-6M. (1972). Bucharest: Selanetoara.
[6] Chebotarev, V.P., Medvedev, A.L., Salapura, Yu.L., & Zubenko, D.V. (2012). Analysis of vertical switchgears of pneumatic seeders. *Mechanization and Electrification of the Rural State: Inter-Industrial Thematic Scientific Collection*, 96, 67-75.
[7] Alt, V.V., Shchukin, S.G., & Valkov, V.A. (2008). The concept of development of sowing machines. *Achievements of Science and Technology of the Agroindustrial Complex*, 9, 44-48.
[8] Yasir, S.H., Liao, Q., Yu, J., & He, D. (2012). Design and test of a pneumatic precision metering device for wheat. *Agricultural Engineering International: CIGR Journal*, 14(1), 16-25.
[9] Starodubtsev, Yu.V., & Solodov, V.G. (2005). Numerical model of viscous turbulent flow in one stage gas turbine. *Journal of Turbomachinery*, 128, 60-67.
[10] Babu, V. (2021). *Fundamentals of gas dynamics* (2nd ed.). Berlin: Springer.
[11] Anderson, J.D.Jr. (1995). *Computational fluid dynamics: The basics with applications*. New York: McGraw-Hill.
[12] Köhler, M. (2011). *Development and implementation of a method for solving the laminar boundary layer equations in airfoil flows*. Darmstadt: Technische Universitat Darmstadt SLA.
[13] Anderson, J.D.Jr. (2017). *Fundamentals of aerodynamics* (6th ed.). New York: McGraw-Hill Education.
[14] Cebeci, T. (2004). *Turbulence models and their applications*. Berlin: Springer.

[15] Chapman, D.R. (1980). Computational aerodynamics and prospects for its development. *Dryden Lecture. Rocket Technology and Cosmonautics*, 18(2), 3-30.
[16] Frank, M. (2016). *White fluid mechanics* (8th ed.). New York: McGraw-Hill Education.
[17] Chung, T.J. (2002). *Computational fluid dynamics*. Cambridge: Cambridge University Press.
[18] Menter, F., & Egorov, J. (2005). Turbulence models based on the length-scale equation. *Fourth International Symposium on Turbulence and Shear Flow Phenomena*, 4, 941-946.
[19] Frost, W., & Moulden, T. (Eds.). (1980). *Turbulence. Principles and applications*. Kyiv: Znannia.
[20] Hirsch, C. (2007). *Numerical computation of internal and external flows* (2nd ed.). Oxford: Published by John Wiley & Sons, Ltd.
[21] Reddy, J.N. (2006). *An introduction to the finite element method*. New York: McGraw-Hill.
[22] Hanlon, P.C. (2001). *Compressor handbook*. Washington: McGraw-Hill.
[23] Solodov, V.G. (2011). *Modeling of turbulence. Calculation of large vortices*. Kharkiv: KHNADU.
[24] ANSYS CFX – Solver theory guide. Release 15.0. (2013). Canonsburg: ANSYS Inc.
[25] Tokushev, M.Kh., & Martych, S.P. (2019). Classification of switchgears of pneumatic seeders. In *Seifullin readings – 15: Youth, science, technology – new ideas and perspectives: Scientific and theoretical conference* (pp. 208-211). Nur-Sultan: Kazakh Agrotechnical University. S. Seifullina.
[26] Prandtl, L. (1904). Über Flüssigkeitsbewegung bei sehr kleiner Reibung. Verhandl III. *Intern. Math. Kongr. Heidelberg*, 2, 484-491.
[27] Fletcher, A.J. (1988). *Computational techniques for fluid dynamics*. New York: Springer-Verlag: Berlin.
[28] Wilcox, D.C. (2006). *Turbulence modeling for CFD* (3rd ed.). La Cañada Flintridge: DCW Industries.
[29] Kornev, N. (2013). *Mathematical modeling of turbulent flows*. Rostock: University of Rostock.
[30] Ronald, H. (2000). *Aungier Centrifugal compressors a strategy for aerodynamic design and analysis*. New York: Asme press.
[31] Lokshin, I.L. (1959). Investigation of the flow behind the wheels of centrifugal fans in relative motion. *Industrial Aerodynamics*, 12, 2-153.

Віктор Іванович Мельник, Олексій Петрович Зеленський, Андрій Петрович Зеленський

Державний біотехнологічний університет
61002, вул. Алчевських, 44, м. Харків, Україна

Численне моделювання газодинамічних процесів у відцентровому радіальному вентиляторі посівних машин

Анотація. Підвищення ефективності рослинництва, де значне місце займають просапні культури (такі, як кукурудза, соняшник, соя, сорго, цукрові буряки та ін.), нерозривно пов’язане з впровадженням енерго- та ресурсозберігаючих технологій для забезпечення якісної та своєчасної реалізації технологічних процесів виробництва. Важливим технологічним процесом є посів. Для підвищення швидкості, як наслідок, продуктивності та, головне, якості проведення польових робіт в даний час застосовуються сівалки точного висіву з пневматичними системами з покращеними характеристиками генераторів вакууму. При цьому вакуумні системи таких посівних машин найчастіше базуються на використанні радіального відцентрового вентилятора. Тому важливим завданням є проектування вентилятора з урахуванням гармонізації його параметрів з параметрами сівалки в цілому вимагає проведення великої кількості випробувань та перевірок. При використанні тільки натурних моделей (стендів) необхідні дуже великі ресурсні та часові витрати. За рахунок чисельного моделювання газодинамічних процесів стає можливим суттєво скоротити кількість натурних експериментів, швидше визначити раціональний вигляд відцентрового радіального вентилятора. Мета даної роботи – розглянути методику застосування чисельного моделювання газодинамічних процесів у відцентровому радіальному вентиляторі за допомогою Computer-aided engineering (CAE) – пакетів. Застосування Computational fluid dynamics (CFD) – методів для дискретизації рішень диференціальних рівнянь з урахуванням рівнянь Ейлера та рівнянь Навье-Стокса. Для вивчення вихрових потоків, що виникають у міжлопаткових каналах вентилятора, використовується чисельне моделювання турбулентності на основі k-ε моделі, k-ω моделі та Shear Stress Transport (SST) моделі. Наведено результати розрахунку тривимірного поля швидкостей та тисків у робочій області вентилятора, визначено раціональні геометричні характеристики, аеродинамічна характеристика (залежність перепаду тиску від витрати повітря). Такий підхід проектування відцентрового вентилятора дає можливість суттєво гармонізувати його параметри на етапі розробки та скоротити час на реалізацію нових проектів

Ключові слова: числове моделювання, k-ε модель, k-ω модель, SST модель, метод кінцевих елементів, метод кінцевих об’ємів, CAD редактор