Journal of Engineering Physics and Thermophysics, Vol. 83, No. 5, 2010

MODELING OF THE OPERATION OF A MINE COUNTERROTATION FAN BY MEANS OF THE FLUENT SUITE

E. I. Gurina

UDC 532.516

Mathematical modeling of the processes in the flow part of an axial counterrotation fan has been carried out by means of the FLUENT suite. Distributions of the gas-dynamic parameters characterizing the basic laws of the investigated process such as air flow velocity, mass flow rate, and air flow pressure have been obtained and dead regions of the construction have been revealed. It has been shown that mathematical modeling can be used for virtual (being developed) models of fans which are further embodied in real objects. The experimental data have been compared with the results of the computational modeling.

Keywords: *mathematical modeling, finite-difference scheme, axial fan, aerodynamic characteristics.*

Introduction. Until recently, to make sure that a product operated efficiently, it was necessary to create a prototype and a test bench for testing its parameters, whose cost exceeded several times the cost of the product being tested. It was at the test stage that the chief limitations of the model design were revealed. For the prototype to be turned into an end product, it is necessary to test dozens of variations of such designs.

In view of the new technologies used in extractive industries, fans with high indices for the pressure and flow rate of air with a lower power consumption are in demand. These are primarily counterrotation fans. However, from the point of view of the processes in them they are complex and difficult to calculate by traditional methods [1].

In the present work, the operation of a mine counterrotation fan is modeled with the use of the FLUENT suite and the accuracy of the obtained solution is demonstrated by comparing it with the experiment: the values of the total differential pressure and the air flow rate are estimated.

The VVM-7 counterrotation fan was designed for ventilating dead-end workings of mines. The model consists of a housing (cylindrical channel), built-in electric motors, two impellers (counterrotating), inlet guide and directing vanes, a collector, and slides. Figure 1 shows the fan as a real physical object (Fig. 1a) and its solid-state model (Fig. 1b). In the problem, it is required to determine the total differential pressure and its corresponding air flow rate for given operating conditions of the fan.

To model the device, it is necessary to describe mathematically the physical processes proceeding during its operation, solve the obtained mathematical problem by a numerical method, and analyze the results. Let us consider this process in more detail using the VVM-7 mine counterrotation fan as the example.

Mathematical Model. The mathematical model describing the air flow in the flow part of the fan is represented by a system of differential equations incorporating Navier–Stokes equations of motion and a continuity equation [2, 3], as well as equations of the turbulence model [4].

The choice of the turbulence model depends on the character of the turbulent flow, the required accuracy, the available computational resources, and the time expenditure reguired for modeling the process. Depending on the suitable turbulence model, the equations forming it are solved (for the given problem, we used the standard k–ε model of turbulence [4] based on the equations of transfer of turbulent kinetic energy k and the dissipation rate ε).

Solution Methods for the Mathematical Problem. To simulate the operation of the fan, a model of its simplest allowable design is constructed. To perform mathematical modeling, there is no need to take into account all technological elements of the flow part. Figure 1b shows a model with a simplified design of the fan. The computational model incorporates such important elements for mathematical modeling as inlet guide vanes (IGV), impellers (I1

Tomsk State University, 36 Lenin Ave., Tomsk, 634050, Russia; email: dragon@temz.tomsk.ru. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 83, No. 5, pp. 924–929, September–October, 2010. Original article submitted December 16, 2008; revision submitted December 18, 2009.

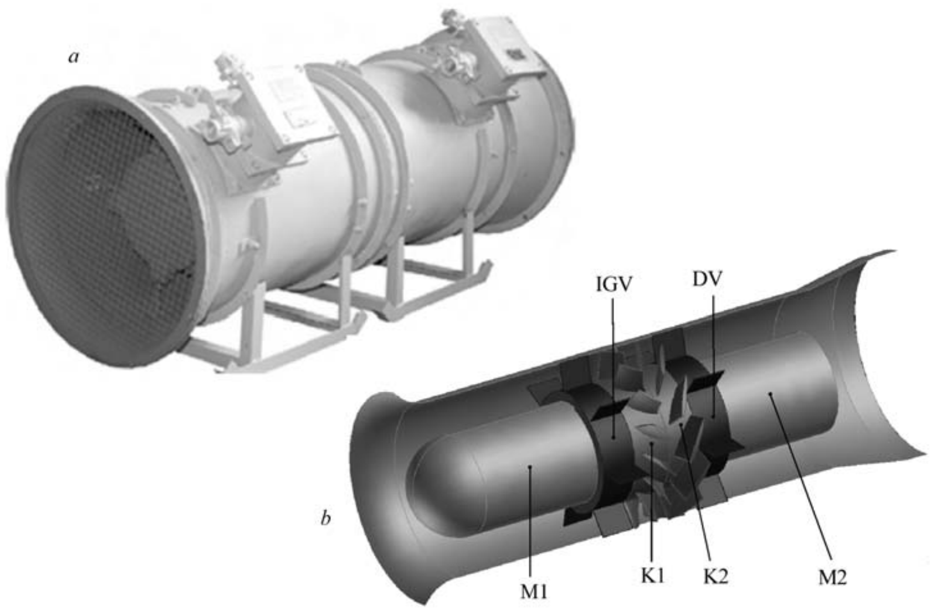

Fig. 1. Models of the VVM-7 fan: a) physical, b) solid-state (M1, M2, electric motors); IGV, inlet guide vane; I1, I2, impellers; DV, directing vane).

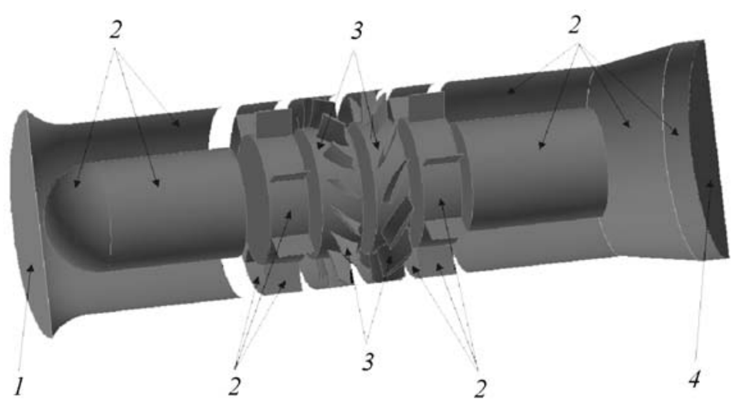

Fig. 2. Splitting of the calculation model into subregions with surfaces chosen for specifying the boundary conditions: 1) surface of the liquid flow into the calculation region; 2) fixed wall; 3) moving wall; 4) surface of the liquid outflow from the calculation region.

and I2), directing vanes (DV), and two motors (M1 and M2). The given model contains both rotating elements (impellers) and stationary ones (inlet guide and directing vanes). On the basis of the geometric configurations of the fan, the flow part of the model is subdivided into subregions on which further in the user FLUENT suite preprocessor a finite-difference mesh is constructed. It is expedient to subdivide the given computational model into six regions: M1, IGV, I1, I2, DV, and M2 (Fig. 1b). In Fig. 2, for visual perception, all subregions are separated by some distance from one another. The main means of solving engineering problems connected with a viscous fluid flow is the application of numerical methods. The most popular discretization method used in solving problems of computational hydrogasdynamics is the finite-volume method. The process of preparing the problem for solution contains the following basic points: construction of the geometry of the object; decomposition of the calculation region into subregions; construction of the finite-difference mesh, as well as specifying the initial and boundary conditions.

To describe the flow field, the flow part of the fan is split into a finite set of control volumes (calculation cells) filling that part of the fan where air is moving.

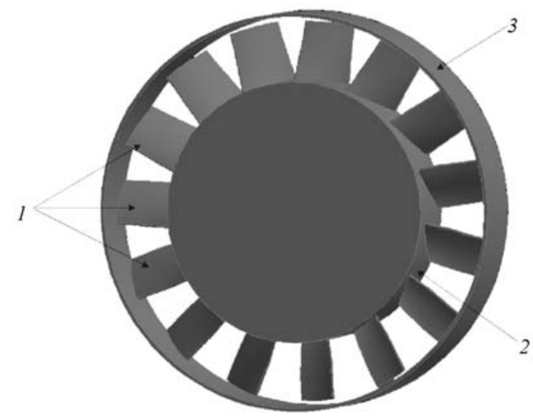

Fig. 3. Configuration of the first impeller blading: 1) blades; 2) hub; 3) housing.

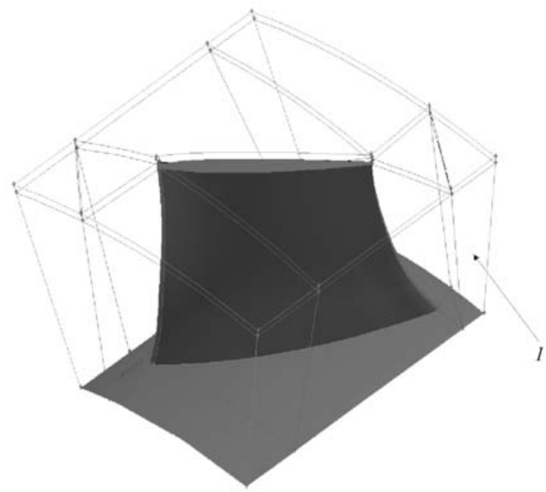

Fig. 4. Decomposition of the periodic channel: 1) one of the 16 volumes surrounding the impeller blade.

Consider the process of constructing a mesh for the blading of the first impeller (I1). To model the behavior of the air flow in the region of the impeller, it is not necessary to construct a mesh for the entire flow zone of the blading (Fig. 3). It is enough to construct a mesh on a chosen periodic channel (surrounding one of the impeller vanes) with an indication of the number of periodic channels (number of vanes) and the selection of periodic boundaries in the channel. Such an approach markedly decreases the resources consumed and shortens the time spent in simulating the process. Figure 3 shows the general configuration of a turbomachine consisting of three basic components such as the vane, the impeller hub, and the housing. Figure 4 shows a periodic channel incorporating an impeller vane, which serves to guide the flow. The lower and upper radial boundaries of the channel are the hub and the housing, respectively. The periodic angle of the channel is 24° (15 vanes). As a rule, the periodic channel is first broken up into large components (its decomposition is carried out) for the convenience of constructing a structured mesh on them. For the given example, the channel was decomposed into 16 geometric volumes, which, in the aggregate, formed the flow region around a vane.

The construction of a mesh on a periodic channel begins with the creation of a boundary layer near the vane surfaces to increase the mesh density near the walls and decrease it at a distance from them. Figure 5 shows the view of the mesh on the vane profile upon breaking of its edges by the mesh nodes. For simplicity of perception, the boundary-layer thickness is enlarged and the crowding of points is absent. Then all faces (surfaces) of the volumes are broken by the structured mesh. Figure 6 shows the view of the finite-difference computational mesh around one of the impeller vanes (the number of control volumes was 357,510). The procedure of constructing a computational mesh for the other components of the fan (IGV, I2, M1, M2) is analogous.

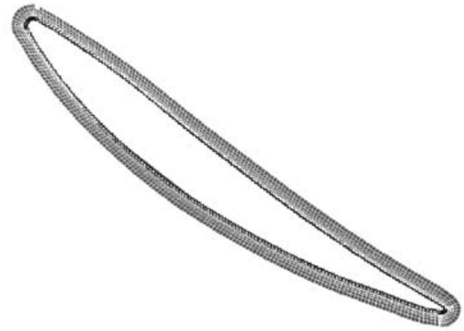

Fig. 5. View of the mesh near the blade profile.

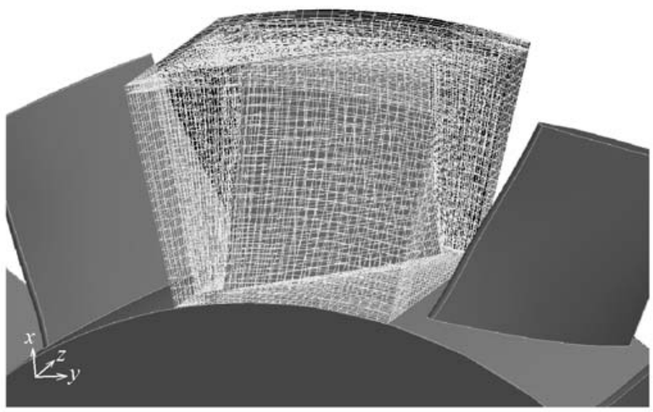

Fig. 6. Periodic segment of the mesh around the I1 blade.

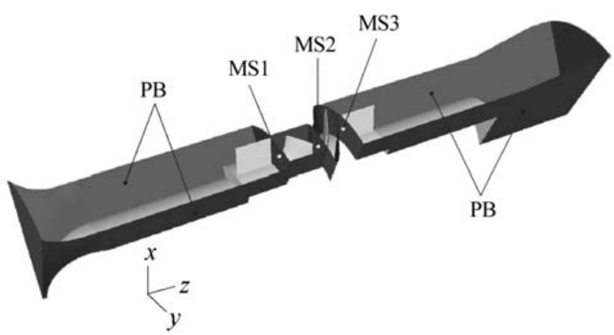

Fig. 7. Periodic segment of the calculation region (PB — periodic boundaries, MS — pairs of mixing surfaces).

The meshes constructed for each subregion are connected. A model for their interaction with one another is determined with the aim of obtaining a single calculation domain. The boundary conditions are determined (according to the type) and specified also on the surfaces of the computational model.

As applied to our problem, we shall specify the boundary conditions on the following surfaces (see Fig. 2): 1) the surface of liquid flow into the computational region where the value of the total pressure P_{tot} = 101,325 Pa is given; 2) the fixed wall, pertaining to the vanes and hubs of the IGV and DV; 3) the moving wall, pertaining to the vanes and hubs of the impellers; 4) the surface of liquid outflow from the computational region where the value of the statistical pressure P_s = 102,500 Pa is given.

The subregions of the impellers I1 and I2 (Fig. 2) rotate with a constant angular velocity ω_1 = –3000 rpm and ω_2 = 3000 rpm, respectively. The surfaces of the vanes and hubs rotate with the angular velocity of their corresponding subregions. In the case of modeling a periodic sector of a subregion, in the mathematical formulation the so-

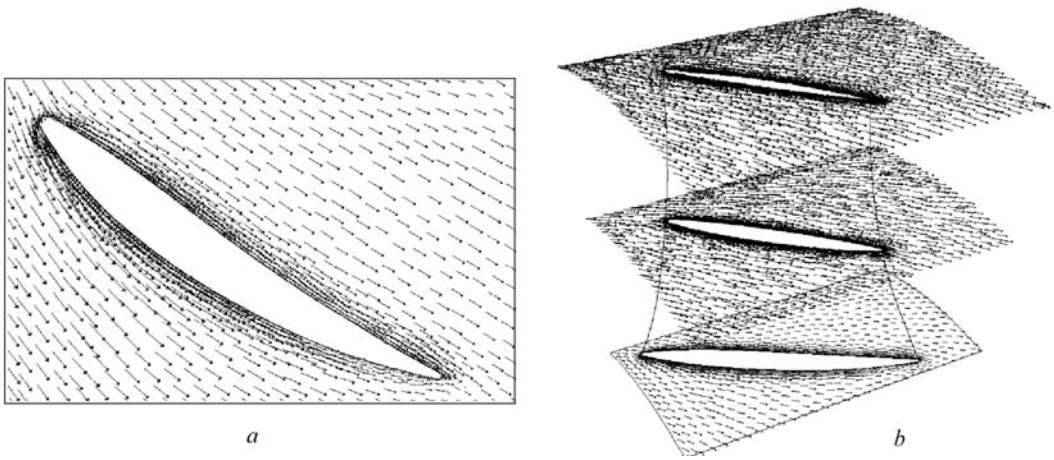

Fig. 8. Relative velocity vectors in the region of I1: a) in the radial cross section of the blade root; b) in three radial cross sections on the blade height.

TABLE 1. Comparison of the Calculated and Experimental Data for Parameters P_{tot} and Q_v

Experiment number	Total pressure P_{tot}, Pa			Air flow rate Q_v, m^3/s		
	Fluent	experiment	Δ, %	Fluent	experiment	Δ, %
1	1627.23	1573.78	3.39	12.76	12.03	6.05
2	2374.76	2305.62	2.99	12.45	11.81	5.40
3	4403.42	4261.02	3.34	11.37	10.77	5.58
4	4715.31	4859.85	2.97	11.06	10.49	5.42
5	5990.25	6042.48	0.86	9.97	9.56	4.33

called periodic boundary PB is considered (see Fig. 7). For the interaction of the rotating and stationary subregions with one another (as well as for the counterrotating ones), the model of mixing of surfaces of [5] is considered. Its main idea is that the parameters of the flow between the subregions combine at the place where the surface of outflow from one subregion and the surface of flow into another subregion are situated. Figure 7 shows three pairs of mixing surfaces (MS1, MS2, MS3) for the interaction of six subregions with one another. For example, the pair MS1 incorporates the surface of outflow of the liquid from the IGV subregion and the surface of flow of the liquid into the I1 subregion. On the outflow surface, the flow data are averaged in the circumferential direction and recorded in a file that will contain the values of such parameters as P_{tot}, k, ε, radial v_r, tangential v_φ, and axial v_z components of the flow velocity. This file is then used as the boundary condition on the surface of flow into the I1 subregion. In turn, on the surface of inflow of the liquid into the I1 subregion, the flow parameters are also averaged and recorded in the data file. The values of the following parameters are recorded: P_s, k, ε, v_r, v_φ, and v_z, and this file is then used as the boundary condition on the surface of outflow of the liquid from the IGV subregion. Thus, the flow information between subregions is combined at the place where the pair MS1 is situated. As a result, in the flow part of the fan the Reynolds-averaged Navier–Stokes equations (for incompressible flow) for the velocity and pressure, as well as the equations for the k–ε standard turbulence model are calculated. The working substance is incompressible air of density $\rho = 1.205$ kg/m^3.

Analysis of the Obtained Results of Calculations. For the analysis of the flow, the obtained results of the calculation are given in the form of velocity fields (in vector form) in several radial cross sections of the model. Figure 8a shows the vector field of the relative velocity in the radial cross section of the model near the I1 hub. Figure 8b gives the velocity vectors in three cross sections on the vane height to obtain a complete pattern of the flow behavior in the intervane channel and near the walls. As is seen from the figure, the flow past the vane surfaces is without separation. In the present paper, visualization of the flow only in the region of the first impeller is given. Note that under the given operational conditions of the fan, flow separation from the vanes of the second impeller is also absent.

The calculation was carried out by the known-in-advance boundary condition at the outlet from the fan P_s (parameter values close to the experimental ones were taken). For the calculation parameters, we used the air flow rate Q_v and the value of the total pressure P_{tot}. Table 1 presents the calculation parameters and the experimental data for P_{tot} and Q_v.

The results of the modeling are in good agreement with the experimental data with account for the fact that the calculation was performed not on a detailed, but on a simplified geometry of the fan. So, upon full-scale tests the discrepancy between the parameter values was within 0.86–3.39% for the total pressure and 4.33–6.05% for the air flow rate.

It should be noted that in the case where it is necessary to obtain more accurate results, the calculation should be made on the basis of a geometry maximally approaching the real one. Undoubtedly, the problem is more difficult in the absence of experimental data, as well as of a real model of the facility (development stage). In this case, to model the operation of the fan under certain conditions (e.g., in the regime of maximum efficiency), we have to calculate all previous regimes along the characteristic dependence of Q_v on P_s or on P_{tot} for a given sample, which takes much more time. Note that for primary evaluation of the potentialities of the design there is no need to perform a detailed calculation, and upon obtaining a general idea of the operation of a machine with a simplified geometric configuration the conditions of interest can be calculated in more detail.

NOTATION

k, turbulent kinetic energy, m^2/s^2; P, pressure, Pa; Q, air flow rate, m^3/s; v_z, v_r, v_φ, axial, radial, and tangential velocity components, m/s; Δ, error in calculations compared to experimental data, %; ε, dissipation rate of turbulent energy, m^2/s^3; ρ, density, kg/m^3; ω, rotation frequency of the impeller, rpm. Subscripts: 1, 2, first and second impellers, respectively; s, static; tot, total; v, volume.

REFERENCES

1. K. A. Ushakov, I. V. Brusilovskii, and A. R. Bushel', *Aerodynamics of Axial Fans and Elements of Their Construction* [in Russian], Gosgortekhizdat, Moscow (1960).
2. H. Schlichting, *Boundary-Layer Theory* [Russian translation], IL, Moscow (1956).
3. L. G. Loitsyanskii, *Mechanics of Liquids and Gases* [in Russian], Drofa, Moscow (2003).
4. W. C. Reynolds and T. Sebesi, *Calculation of Turbulent Flows* [Russian translation], Mashinostroenie, Moscow (1980).
5. FLUENT. *User's Guide*, 2008.