Journal of Engineering Physics and Thermophysics, Vol. 71, No. 5, 1998

CALCULATION OF THE WIND ACTION ON A COOLING TOWER IN THE PRESENCE OF DEVICES FOR EQUALIZING THE GROUND FLOW

V. V. Bungov, S. A. Isaev, and V. B. Kharchenko

UDC 532.517.4

Based on a solution of the Reynolds equations closed using a two-parameter dissipative turbulence model by the finite-volume method, the concept of equalizing the wind flow in the lower part of a cooling tower upon placing shield barriers in its vicinity is substantiated.

1. Among numerous problems of finding and substantiating rational technical solutions in the operation of various power engineering structures, including, e.g., cooling towers, the problem of numerical modeling of the effect of the spatial ground wind flow on these structures occupies an important place. The practical value of the solution of the problem consists in determination of the most efficient approaches to the control of the flow near the structure, in particular, in the vicinity of the windows of a cooling tower, in order to avoid unfavorable regimes of water atomizing that yield icing in the form of distinctive hummocks in wintertime. The efficiency of the control is evaluated by the degree of decrease in the wind flow intensity near the windows of the cooling tower, and setting up shield barriers in the vicinity of the cooling tower is chosen as a means to achieve the objective. Thus, the solution of the problem is found within the framework of the promising aerohydrodynamic approach connected with providing streamlining of structures having various functions by forced formation of large-scale vortex structures in their vicinity [1].

As a whole, the specified problem of the interaction of the wind flow with the cooling tower belongs to the class of stationary spatial problems, and the regime of completely developed turbulent flow realized in this case is characterized by a substantial mutual effect of structural features connected with deformation of the boundary flow near the windows and formation of spatial stall zones and vortex filaments behind the structure and the positioned shield barriers. Therefore, to solve this problem, we use a numerical approach developed in [2, 3] that is based on consideration of the complete Reynolds equations closed using a two-parameter dissipative turbulence model. In this work, much attention is given to developing curvilinear spatial grids matched to the surface of the body streamlined. Here, preference is given to algebraic grids in order to achieve on-line and efficient operation of the computational complex as a whole. The graphical interpretation of the information obtained upon solving spatial problems required development of dedicated software that is a component of the computational complex, which also has independent significance due to the universal character of its application to problems of various types. Testing of the numerical algorithm was carried out on a three-dimensional problem of streamlining of an Akhmed model body for which reliable experimental data are available [4, 5].

2. Until recently, problems of the aerodynamics of power engineering structures of various designs and purposes were solved mainly within the framework of the experimental approach to investigations, including full-scale tests. Only certain comparatively simple flow regimes for substantially simplified configurations of the objects under investigation could be predicted numerically. Here, during the modeling, assumptions substantially simplifying the formulation of the problems were introduced, the strongest of which were neglecting viscous effects and restricting the dimensions of the problems solved to the case of two-dimensional flows. At the same time, it is evident that a modeling complex adequate for the physical object should provide calculations of spatial flows in

Civil Aviation Academy, St. Petersburg, Russia. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 71, No. 5, pp. 866-871, September-October, 1998. Original article submitted February 27, 1997.

 1062-0125/98/7105-0852$20.00

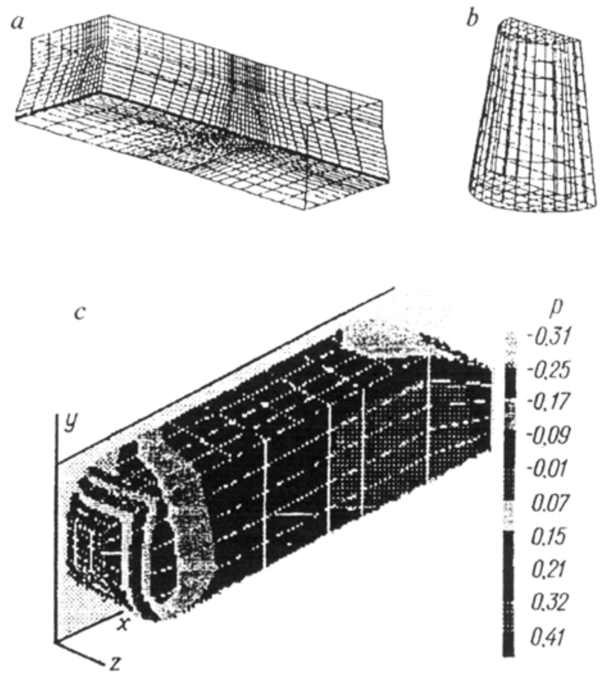

Fig. 1. Fragment of a grid for calculations in the vicinity of a cooling tower (a) and on its surface (b) and isobaric zones on the surface of an Akhmed body streamlined by a low-velocity flow (c).

curvilinearly shaped regions with the necessary account for both viscous effects and the possible complex character of the flow arising.

In the present investigation, for a given geometry of the cooling-tower model, we present calculations of the stationary ground flow developing upon the action of the wind flow on the object under consideration. Devices for equalizing the aerodynamic characteristics are placed in the vicinity of the windows of the cooling tower to decrease the effect of the wind on the flow of the cooling liquid. The problem is solved in a simplified formulation by neglecting the motion of the cooling liquid (i.e., without account for blocking or penetrability of the windows). We also neglect the mechanism of the natural convection in the cooling tower itself, and the flow within it is created by a model of a ventilator installed in the middle part of the cooling tower. Thus, we consider low-velocity motion of air with prevailing convective transfer with account for the effect of turbulent diffusion within the framework of a dissipative two-parameter turbulence model. Among the justified assumptions for solving the given problem is the assumption of a symmetric character of the flow with respect to the symmetry plane passing through the geometric axis of the cooling tower parallel to the incident wind flow. As a result, a substantial economy of computational resources is achieved without sacrifice of the quality of the numerical predictions obtained. The calculations region presented in Fig. 1a is bounded by control surfaces situated at a substantial distance from the object under investigation. Strict conditions on the incident-flow velocity are set up on the left boundary. On other boundaries, open to the air flow, so-called "mild" boundary conditions are imposed, which are realized upon extrapolation of the flow parameters from within the region of calculations to the boundary. On solid impenetrable walls, including the inner and outer surface of the cooling tower (the thickness of the wall of the cooling tower is negligible – see Fig. 1b), boundary functions widely used in solving practical engineering problems are specified. In this case, the system of original equations includes the continuity equation and the equations of motions in the Reynolds approximation for the Cartesian components of the flow velocity written in generalized curvilinear coordinates and supplemented with a pair of equations of a two-parameter dissipative turbulence model for the energy of turbulent pulsations and the rate of its dissipation. The density of the air, the velocity of the air wind flow, and the maximum diameter of the cooling tower typical of the problem under consideration are chosen as the nondimensionalizing parameters. The following variants are considered: 1) numerical simulation of the flow in the cooling tower in the absence of wind (in this case, a dimensionless velocity with a value of 0.19 times the wind flow velocity, which is close to the actual value, is maintained in the middle cross section of the cooling tower), 2) simulation of streamlining of the cooling tower by a wind flow in the absence of a device for equalizing the characteristics of the flow in the vicinity of the windows, and 3) in the presence of such a device. A rectangular fence (for the chosen N-shaped structure of the grid for the calculations) with a height equal to that of a window is used as this device. The Reynolds number in the calculations performed was taken equal to $1.65 \cdot 10^6$.

3. In discretization of the problems and development of an algorithm, the substantial nonlinearity of the original equations and the presence of a small parameter at the highest derivative result in certain problems with the stability of the numerical procedure and the numerical diffusion. Approximation of the convective terms of the transport equations is a key point in these problems. As has been shown in detailed test investigations [1, 2], the Leonard quadratic counterflow scheme (its one-dimensional analog) which makes it possible to minimize the distorting effect of the artificial diffusion on the solution of the problems, especially in the case where flows with a complex hydrodynamic structure are considered, is efficient in solving multidimensional problems. At the same time, stability of the numerical procedure is attained by application of single-sided counterflow approximations in the scheme written for the increments of the dependent variables and by introducing an additional diffusion term, with the diffusion coefficient evaluated in numerical experiments, into the implicit part. High efficiency of the numerical procedure is provided by using curvilinear grids with nondisplaced nodes in combination with introduction of a monotonizer into the pressure correction unit within the framework of the Rhi–Chou approach and by using the Buleev incomplete matrix factorization method (known in the foreign literature as Stone's SIP method) in solving systems of algebraic equations. In general, the concept of splitting of physical processes is implemented within the SIMPLEC procedure. It should be noted that introducing a pseudo-nonstationary term into the implicit part of the original equations leads to an increase in the stability of the numerical procedure and expands the possibilities of control of the computational process.

The expenditure of time for the calculations is determined to a high degree by the type of grid employed. It is evident that an optimum grid structure is characterized by a minimum number of distributed calculation nodes. The software package developed implements calculation, construction, and graphical presentation of the grid for calculations in an interactive mode for an arbitrary set of initial parameters. The construction of the grid for the problem under consideration is carried out within an algebraic approach to the determination of the positions of the calculation nodes. In the case under consideration, an N-type grid with the number of nodes of order of 10^4 that re unevenly distributed over the region calculations and are more dense near the surface of the cooling tower ($33 \times 21 \times 14$) is used. Fragments of the grid are presented in Fig. 1a and b. The height of the cooling tower, the diameter of the upper cross section, and the height of the windows were equal to 2.4, 0.65, and 0.15, respectively, in fractions of the cooling-tower diameter at the level of the windows.

The reliability of the numerical results obtained is the subject of a special numerical investigation. It is desirable to compare the results with experimental data on each of the problems under consideration. However, since these data are absent, we test the numerical algorithm itself, i.e., its main structural elements. As a rule, in investigations of this type, one chooses a problem with a simplified formulation for which a physical analog and reliable experimental data exist. At the same time, the test problem chosen should reflect the special features of the flows under consideration.

To do this, one can use the problem of stationary spatial streamlining of a model body whose configuration has been proposed by Akhmed [5]. The body is a parallelepiped with rounded sharp edges in the front part and a tapered wedge-shaped tail part that is situated near a movable screen mimicking a road bed. Some results of calculations in Fig. 1c for a Reynolds number equal to $3.6 \cdot 10^6$, corresponding to the conditions of tests of the model in a wind tunnel, illustrate the pronounced asymmetric distribution of local force loads on the object under consideration. According to experimental data [6], the head-drag coefficient of the body lies within the limits of 0.15 to 0.17. The value of the head-drag coefficient calculated on a grid consisting of approximately 10^4 cells equals 0.17, which gives an idea of the reliability of numerical predictions of spatial flows by means of the developed computational complex on comparatively economical grids. It should be noted that the data obtained correlate well with results of calculations of streamlining of the Akhmed body on more detailed grids using the well-known FLOW3D software package [6].

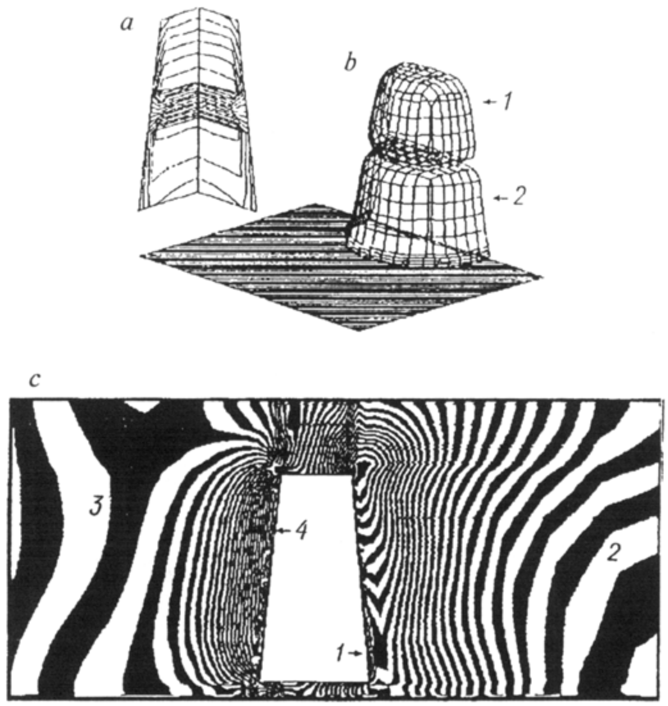

Fig. 2. Isobars from −0.07 to 0.09 with a step of 0.01 in a cross-shaped cross section (a) and spatial isobaric surfaces (b) [1) $p = 0.01$, 2) -0.01] within the cooling tower in the absence of wind flow; isobar pattern in the symmetry plane in the case of streamlining of the cooling tower by a wind flow (c) [1) $p = -0.3$, 2) -0.1, 3) 0.1, 4) 0.25]. p is the excess pressure expressed in fractions of the doubled velocity pressure of the wind.

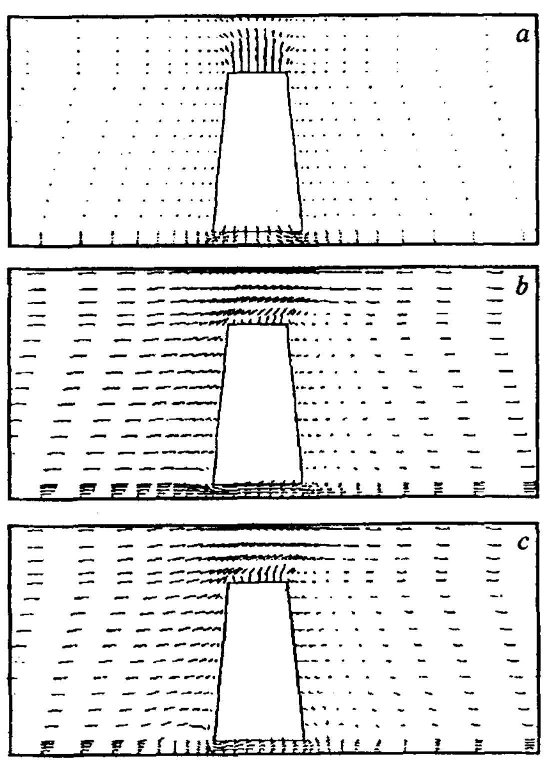

Fig. 3. Field of velocity vectors in the symmetry plane in the vicinity of a cooling tower in the absence of wind (a), under wind action (b), and with an equalizing device positioned (c).

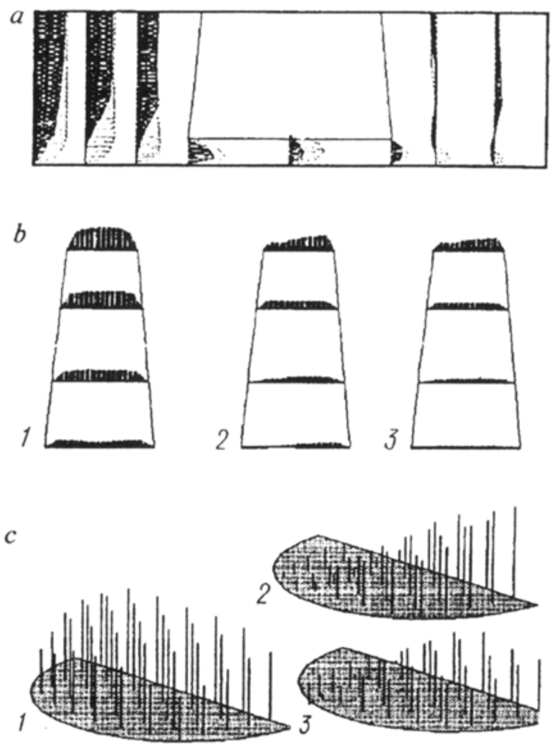

Fig. 4. Air flow velocity profiles near a cooling tower in the symmetry plane in the absence and in the presence of a shield fence (a); profiles of the vertical velocity component in the central (b) and lower (at the window height) (c) cross section of the cooling tower (1, 2, and 3 are patterns without wind, with wind action, and in the presence of a shield fence, respectively).

4. The results presented below were obtained by means of the above-described software package for automated graphical processing of numerical information obtained for spatial flows. Figures 2-4 present results of numerical simulation of the flow in the vicinity of the cooling tower in different situations: without a wind flow, with a wind flow but without a device equalizing the flow near the windows, and, finally, with a wind flow in the presence of an equalizing device in the form of a fence surrounding the cooling tower and shading the windows (a fence with a height equal to that of a window is situated at a distance of 0.5 from the window).

Solution of the problem of the flow in the vicinity of the cooling tower without wind action was carried out not only to determine the character of the flow realized in this case but also to adjust the computational complex to substantiate the technical solution. As follows from the results of numerical calculations presented in Figs. 2-4, the flow in the space surrounding the cooling tower that is induced by the motion of the air flow within the tower has a completely axisymmetric character. Here a virtually runaway channel-type flow with a steady pressure drop along the tower axis with increasing height is established within the cooling tower (Fig. 2a). The velocities of the air flow are not high in the vicinity of the windows, and the profile of the vertical component of the velocity is close to uniform in the lower cross section of the cooling tower (Fig. 4b, c). A rather weak flow is induced in the surrounding space, mainly due to the outflow from the upper cross section (Fig. 3a).

The wind action deforms the flow field strongly in the vicinity of the cooling tower. In principle, this is natural, since the specified velocity of the air wind flow exceeds substantially (fivefold) the velocity of the simulated internal flow in the cooling tower (with respect to the velocity in its middle cross section). It is obvious that the key point is the interaction of the ground wind flow with the obstacle on the plane, i.e., the structure under consideration. The flow has a pronounced spatial character with separation behind the cone-shaped body. The negative effect of the wind flow consists in deformation of the flow field in the vicinity of the windows of the cooling tower, which leads, as is well known, to problems in the functioning of this structure. The striving to avoid undesirable consequences, in particular, atomizing of the cooling liquid within the windows, has stimulated the search for means of decreasing the velocity of the side flow in the gap between the cooling tower and the ground. Setting up a fence surrounding the cooling tower and shielding the windows from wind action is considered as one of possible variants. A comparative analysis of results of the wind effect on the cooling tower in the absence and in the presence of a device equalizing the characteristics of the flow under the cooling tower is presented in Figs. 3 and 4. Most of results presented refer to distributions of the flow velocity and the pressure in the symmetry plane, since they characterize the situation rather completely and are simpler to interpret graphically.

As follows from a consideration of the velocity vector patterns in Figs. 3 and 4, setting up a fence substantially deforms the flow field in the ground zone near the cooling tower. The equalizing device decreases the flow intensity under the structure considered (Fig. 4a) and smooths somewhat the profile of the vertical velocity component in the lower cross section of the cooling tower (Fig. 4c). Thus, we substantiate the idea of controlling the wind flow near the cooling tower by placing a row of shield obstacles in its vicinity. It is evident that in the next stage of the investigation it is advisable to carry out optimization of the size and location of the obstacles by means of a multiparameter numerical experiment.

The authors are grateful to Prof. A. S. Ginevskii for fruitful discussions of the problem.

The work was carried out under the support of the Russian Foundation for Basic Research, grants Nos. 95-01-00728 and 98-01-00432.

REFERENCES

1. I. A. Belov, S. A. Isaev, and V. A. Korobkov, Problems and Methods of Calculation of Detached Flows of Incompressible Fluids [in Russian], Leningrad (1989).
2. S. A. Isaev and A. E. Usachev, in: Industrial Aerodynamics [in Russian], Issue 4 (36), Moscow (1991), pp. 43-75.
3. S. A. Isaev, V. B. Kharchenko, and Ya. P. Chudnovskii, Inzh.-Fiz. Zh., 67, Nos. 5-6, 1013-1017 (1991).
4. T. Han, D. C. Hammond, and C. J. Sagi, AIAA J., 30, No. 4, 882-889 (1992).
5. V. Sumantran and D. Hammond, GMR Report 6438, General Motors Research Labs., Warren, MI (1988).
6. A. D. Burn, D. S. Clark, I. P. Jones, S. Simcox, and H. S. Wilkes, in: Computational Fluid Dynamics, Proceedings of the International Symposium, Sydney, 1987, Amsterdam (1988), pp. 315-327.