Journal of Engineering Physics and Thermophysics, Vol. 67, Nos. 5-6, 1994

CALCULATION OF A THREE-DIMENSIONAL FLOW OF A VISCOUS INCOMPRESSIBLE LIQUID IN THE NEIGHBORHOOD OF A SHALLOW WELL ON A FLAT SURFACE*

S. A. Isaev, V. B. Kharchenko, and Ya. P. Chudnovskii

UDC 532.517.4

A three-dimensional flow in the neighborhood of a shallow well on a plane was simulated numerically on the basis of a solution of the Reynolds equations written in curvilinear nonorthogonal coordinates and closed by a two-parametric dissipative model of turbulence.

1. The search for unconventional approaches to intensification of heat and mass transfer processes in combination with reduction of hydrodynamic losses stimulated the use of the promising hydromechanical concept of controlling the flow by formation of large-scale vortex structures in the boundary layer. For example, it is known [1] that in the organization of developed circulation zones a system of plates of different sizes placed on the surface in the flow and a circular disk placed in front of a blunt cylindrical body that is of smaller size than the latter decrease substantially the resistance to the flow. The structure of a large-scale vortex is steady and its size is limited to the developed two-dimensional circulation zone. In particular, in the case of axial symmetry a toroidal vortex is localized in the front separation zone in the cavity formed by a protruding disk and the surface of the blunt body.

Application of an ordered coating with wells to a surface in a flow as well as location of plates near the wall in a certain way can generate large-scale vortices near the wall. Previous experimental studies (see, for example, [2]) showed that it was possible to develop so-called self-organized vortex structures of cyclone type capable of increasing the rate of heat and mass transfer while simultaneously decreasing in the hydrodynamic losses. These structures are clearly three-dimensional and unstable. It is of interest to investigate the mechanism of generation of these structures in the simplified case of a single shallow spherical well on a flat surface.

In the present work emphasis is placed on the development and adjustment of a computation complex for numerical simulation of a three-dimensional flow of an incompressible liquid in the domain of curvilinear geometry. A comparative analysis of the present results and data of previous physical experiments provides high reliability for the computation and a more detailed picture of the processes studied that is based on detailed visualization of the simulated flow.

2. Development of implicit algorithms that are based on the concept of splitting over physical processes and use finite-difference schemes with low numerical diffusion encouraged the design of computation complexes with high efficiency and accuracy of numerical predictions. Extensive tests of semiempirical models of turbulence of gradient type, including, in particular, the two-parametric dissipative model of turbulence, ensure physically adequate results with accuracy sufficient for engineering practice. Moreover, wide use of personal computers such as PC AT 386/387, 486 with a large capacity of the primary memory and a high clock frequency and graphic and work stations (DEC, SUN) allow complicated practical situations to be predicted efficiently and at a reasonably low cost.

At present a number of instruments have been developed for solving three-dimensional problems of aerohydrodynamics and heat transfer, using finite-volume and finite-element methods of computation. They

* The work has been financed by the Russian Basic Research Foundation (Grant No. 94-02-04092 a).

Civil Aviation Academy, St. Petersburg, Russia. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 67, Nos. 5-6, pp. 373-378, November-December, 1994. Original article submitted September 2, 1993.

include the widely known program products PHOENICS, FIDAP, and FLOW3D. Original codes used in the present study are based on solution of steady-state elliptical Navier-Stokes equations written in terms of Cartesian velocity components, Reynolds-number averaged, and closed within the standard two-parametric model of turbulence. In the dimensionless form the initial equations for the generalized variable $\varphi = (u, v, w, k, \varepsilon)$ in the curvilinear nonorthogonal coordinates ξ, η, ζ have the form

$$(U\varphi)_\xi + (V\varphi)_\eta + (W\varphi)_\zeta = \left[\frac{\Gamma^\varphi}{J}(D_{11}^2 + D_{12}^2 + D_{13}^2)\varphi_\xi\right]_\xi +$$

$$+ \left[\frac{\Gamma^\varphi}{J}(D_{21}^2 + D_{22}^2 + D_{23}^2)\varphi_\eta\right]_\eta + \left[\frac{\Gamma^\varphi}{J}(D_{31}^2 + D_{32}^2 + D_{33}^2)\varphi_\zeta\right]_\zeta + S^\varphi J . \quad (1)$$

Here the subscripts ξ, η, ζ denote derivatives along the corresponding coordinate directions. Apart from the source and sink components proper, the source terms S^φ contain the diffusion terms caused by nonorthogonality of the coordinate system chosen. The latter are expressed as follows:

$$S_c^\varphi = \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{11}D_{21} + D_{12}D_{22} + D_{13}D_{23})\varphi_\eta + \frac{\Gamma^\varphi}{J}(D_{11}D_{31} + D_{12}D_{32} + D_{13}D_{33})\varphi_\zeta\right]_\xi +$$

$$+ \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{11}D_{21} + D_{12}D_{22} + D_{13}D_{23})\varphi_\xi + \frac{\Gamma^\varphi}{J}(D_{21}D_{31} + D_{22}D_{32} + D_{23}D_{33})\varphi_\zeta\right]_\eta +$$

$$+ \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{11}D_{31} + D_{12}D_{32} + D_{13}D_{33})\varphi_\zeta + \frac{\Gamma^\varphi}{J}(D_{21}D_{31} + D_{22}D_{32} + D_{23}D_{33})\varphi_\eta\right]_\zeta .$$

The source components determined by the pressure gradient are expressed as

$$S_j^\varphi = \frac{1}{J}(p_\xi D_{1j} + p_\eta D_{2j} + p_\zeta D_{3j}) ,$$

where $j = 1, 2, 3$ correspond to equations for u, v, w. The remaining source terms for the j-th momentum equation are written as

$$S_j^\varphi = \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{11}u_\xi + D_{12}v_\xi + D_{13}w_\xi)D_{1j} + \frac{\Gamma^\varphi}{J}(D_{11}u_\eta + D_{12}v_\eta + \right.$$

$$\left. + D_{13}w_\eta)D_{2j} + \frac{\Gamma^\varphi}{J}(D_{11}u_\zeta + D_{12}v_\zeta + D_{13}w_\zeta)D_{3j}\right]_\xi +$$

$$+ \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{21}u_\xi + D_{22}v_\xi + D_{23}w_\xi)D_{1j} + \right.$$

$$\left. + \frac{\Gamma^\varphi}{J}(D_{21}u_\eta + D_{22}v_\eta + D_{23}w_\eta)D_{2j} + \frac{\Gamma^\varphi}{J}(D_{21}u_\zeta + D_{22}v_\zeta + D_{23}w_\zeta)D_{3j}\right]_\eta +$$

$$+ \frac{1}{J}\left[\frac{\Gamma^\varphi}{J}(D_{31}u_\xi + D_{32}v_\xi + D_{33}w_\xi)D_{1j} + \frac{\Gamma^\varphi}{J}(D_{31}u_\eta + D_{32}v_\eta + D_{33}w_\eta)D_{2j} + \right.$$

$$\left. + \frac{\Gamma^\varphi}{J}(D_{31}u_\zeta + D_{32}v_\zeta + D_{33}w_\zeta)D_{3j}\right]_\zeta .$$

It is important that the above source terms are formulated in divergent form, which allows construction of a conservative computation scheme.

The contavariant variables U, V, W responsible for convective transfer through the faces of the cells are defined by

$$U = D_{11}u + D_{12}v + D_{13}w\,, \quad V = D_{21}u + D_{22}v + D_{23}w\,, \quad W = D_{31}u + D_{32}v + D_{33}w\,.$$

In the above equations the metric coefficients D_{ij} and the Jacobian J are a function of the Cartesian coordinates x, y, z:

$$D_{ij} = \begin{vmatrix} y_\eta z_\zeta - y_\zeta z_\eta & x_\zeta z_\eta - x_\eta z_\zeta & x_\eta y_\zeta - x_\zeta y_\eta \\ y_\zeta z_\xi - y_\xi z_\zeta & x_\xi z_\zeta - x_\zeta y_\xi & x_\zeta y_\xi - x_\xi y_\zeta \\ y_\xi z_\eta - y_\eta z_\xi & x_\eta z_\xi - x_\xi z_\eta & x_\xi y_\eta - x_\eta y_\xi \end{vmatrix},$$

$$J = x_\xi y_\eta z_\zeta + x_\zeta y_\xi z_\eta + x_\eta y_\zeta z_\xi - x_\xi y_\zeta z_\eta - x_\eta y_\zeta z_\xi - x_\zeta y_\eta z_\xi\,.$$

The high Reynolds-number version of the dissipative two-parametric model of turbulence in combination with the method of near-wall functions [1] is used for calculating turbulent flows. This approach is valid for simulation of fully developed turbulent flows.

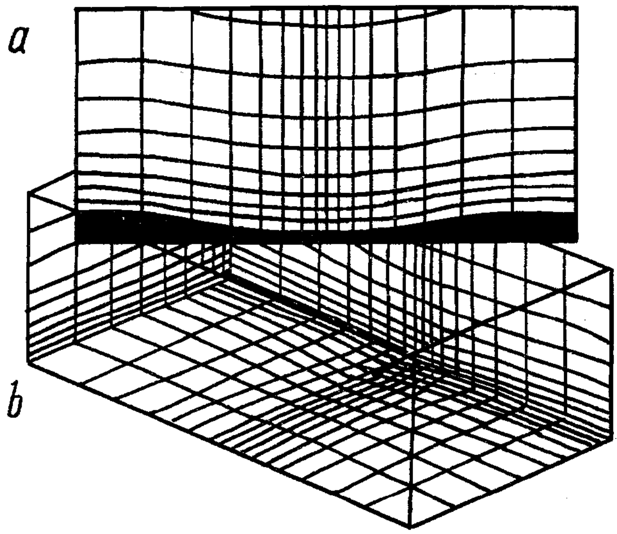
Equations (1) are discretized within the concept of splitting over physical processes, using the finite volume method for equations written in terms of increments of dependent variables in the so-called E-factor formulation [1]. In this case, the Leonard scheme with quadratic interpolation in the counterflow direction and low numerical diffusion is used to approximate the convective terms in the explicit part of the transfer equations so as to obtain high accuracy of the computation when there are regions with high gradients of the parameters in the flow field, in particular, developed separation zones. Meanwhile, to increase the stability of the computation process and to smooth possible nonphysical oscillations in the implicit part of the equations, a counterflow scheme with unilateral differences in combination with introduction of additional artificial diffusion is used. This makes it possible to control the computation process and to achieve more rapid convergence of global iterations.

High efficiency is ensured for computations not just by using of incomplete Boolean matrix factorization to solve systems of difference equations. In the pressure correction unit the SIMPLEC coordinated procedure suggested by Van Durmaal and Reitby is used, which, unlike the Patankar–Spalding SIMPLEC method, prevents relaxation in determination of the pressure correction. The efficiency is also increased by usung the Rhi–Chou method, which consists in introducing a monotonizer for leveling the distribution of the pressure correction into the centered grid without shifted nodes for the velocity components.

The initial-data processing unit contains a generator of curvilinear nonorthogonal grids adapted to the surface in the flow. The computation grids are constructed numerically with automated arrangement of the nodes within the computation region either from solution of systems of elliptical differential equations or by means of an algebraic procedure. The codes developed were tested in solving many two- and three-dimensional problems, including those having physical analogs. Numerical results on adjustment of the complex and estimation of the reliability of numerical predictions for problems on a body in a separating flow and on liquid motion in channels are presented in detail in monograph [1].

A computation of a three-dimensional flow in the neighborhood of a shallow well on a smooth surface was carried out on a PC AT 386/387. In this case the developed software for graphic interpretation of numerical results allowed visualization of three-dimensional flows. The codes used demonstrate universality, efficiency, and robustness for this instrument of numerical investigation.

3. Numerical simulation of turbulent flow in the neighborhood of the well was carried out with the full-scale Reynolds number $\mathrm{Re} = 4.138 \cdot 10^4$, corresponding to a physical experiment in a wind tunnel [3]. The Reynolds number was constructed in terms of the velocity of the main flow (u = 16 m/sec) and the diameter of the well (D = 0.0375 m). These parameters were chosen as normalization scales in this problem. The depth of the well was 0.067.



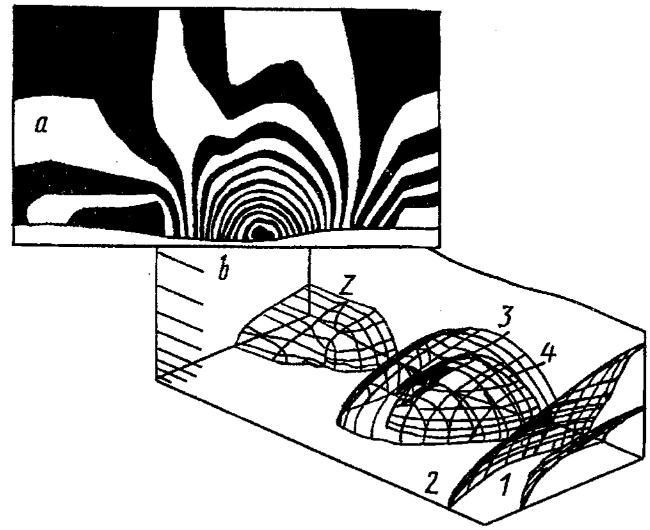
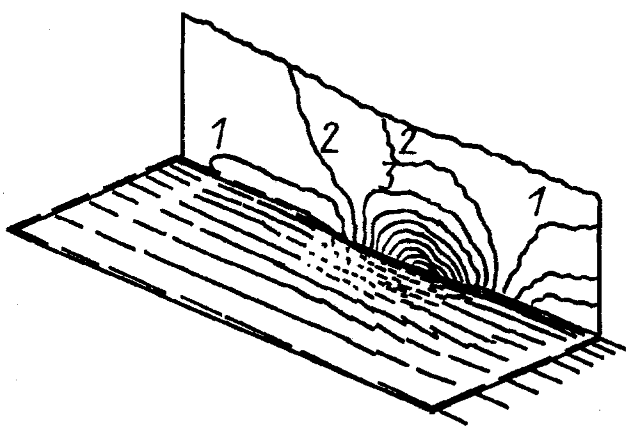


Fig. 1. Fragments of the computation grid in the neigborhood of the well (2) and in the symmetry plane (b).

Fig. 2. Patterns of isobars in the symmetry plane drawn with the step Δp = 0.005 (a) and isobaric surfaces in the environment of the well (b). Digits indicate pressure levels: 1) $p = -0.01$; 2) -0.005; 3) 0.01; 4) 0.02.



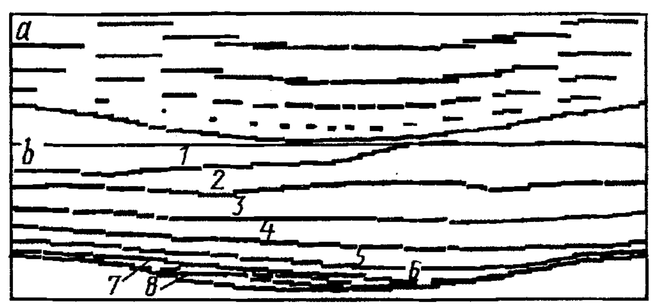


Fig. 3. Patterns of flow velocity vectors in the wall layer together with the pattern of isobars in the symmetry plane drawn with the step $\Delta p = 0.005$: 1) $p = -0.005$; 2) 0; 3) 0.005.

Fig. 4. Patterns of velocity vectors (a) and lines of equal flow velocities (b) in the symmetry plane near the well. Digits indicate corresponding values of the velocity levels: 1) $u = 0.95$; 2) 0.9; 3) 0.75; 4) 0.5; 5) 0.25; 6) 0.1; 7) 0.05; 8) 0.02.

The computation was carried out, using a curvilinear nonorthogonal grid containing $30 \times 16 \times 16$ nodes distributed nonuniformly in the region, being more densely distributed in the neighborhood of the well. The minimum step is 0.027. The flow considered was assumed to be symmetric about the geometric plane of symmetry parallel to the flow direction. The computation region covers the space bounded from below by the concave surface in the flow, on one side by the plane of symmetry, and on the other side by the permeable control plane. The distance between these planes is 3.3. The entrance and exit boundaries are symmetric about the well at a distance of 3.85 and normal to the plane of symmetry. The computation region is bounded from above by a flat permeable control surface located at a distance of 4 from the lower wall. Contours of the computation region with the nodes are shown in Fig. 1a. The location of the nodes in the neighborhood of the well is shown in more detail in Fig. 1b.

The boundary conditions are formulated in the usual way for this type of problem. Parameters of the main flow, which is assumed to be uniform, are prescribed at the entrance boundary. Symmetry conditions are satisfied on the plane of symmetry. Parameters of the flow and characteristics of the turbulence in the neighborhood of the wall are determined by the well-known method of near-wall functions. The so-called mild boundary conditions or conditions of continuation of the solution are formulated on the other control boundaries. The iteration process is continued to convergence determined by achievement of local characteristics of the flow with the required accuracy.

In Figs. 2–4 some of the results obtained are presented. It is interesting that even in the case of a shallow well substantial pressure differences are observed in the simulated flow. As is shown in Fig. 2a, the pressure maximum is shifted from the geometric center of the well to its side exposed to the flow. In the region of increased pressure, domelike isobaric surfaces encompassing this region with pressure monotonically decreasing with increase in the distance from the pressure maximum are formed (Fig. 2b). Regions of lower pressure, where the flow is accelerated locally, occur in front of and behind the well near the surface. Comparison of the computation results with measurements of static pressure on the wall in the flow shows that the maximum pressures and the minimum pressures are very close: $p_{\mathrm{comp}}^{\mathrm{max}} = 0.043$; $p_{\mathrm{exp}}^{\mathrm{max}} = 0.046$; $p_{\mathrm{comp}}^{\mathrm{min}} = -0.020$; $p_{\mathrm{exp}}^{\mathrm{min}} = -0.028$.

Patterns of the velocity vectors in the layer adjacent to the wall (see Figs. 3, 4) indicate substantial changes in the flow in the zone of the well facing the flow. It is in this region that the flow is decelerated and turned toward the symmetry plane. It is especially noteworthy that evolution of the flow is observed only near the wall. Far from the wall the liquid flow is almost unaffected by the existence of the well on the surface in the flow. It is likely that the observed turn of the flow toward the symmetry plane in the presence the well on the flat surface in the flow can be a triggering mechanism for generation of three-dimensional vortex structures in the wall layer near the well.

NOTATION

x, y, z, Cartesion coordinates; ξ, η, ζ, curvilinear nonorthogonal coordinates; u, v, w, Cartesian velocity components; U, V, W, contravariant variables; p, static pressure related to the doubled velocity head of the main flow; k, turbulence energy; ε, dissipation rate of trubulent energy; φ, generalized dependent variable; Γ^{φ}, diffusion transfer coefficient; J, Jacobian; D_{ij}, tensor of metric variables; S^{φ}, source term in the transfer equation; S_j^{φ}, source term in the j-th momentum equation caused by diffusion transfer; S_p^{φ}, source term caused by the pressure gradient; S_c^{φ}, source term related to obliqueness of the coordinate system; Re, Reynolds number. Subscripts and superscripts: max, maximum value; min, minimum value; comp, computed value; exp, experimental value.

REFERENCES

1. I. A. Belov, S. A. Isaev, and V. A. Korobkov, Problems and Calculation Methods of Separated Incompressible Flows [in Russian], Leningrad (1989).
2. V. N. Afanas'ev, P. S. Roganov, and Ya. P. Chudnovskii, Inzh.-Fiz. Zh., **63**, No. 1, 23-27 (1992).
3. V. I. Afanas'ev, V. Yu. Veselkin, A. I. Leont'ev, and Ya. P. Chudnovskii, Hydrodynamics and Heat Transfer in a Flow Past Solitary Wells on an Initially Smooth Surface, Preprint No. 2-91, Pt. 2, N.E. Bauman Moscow State Engineering School, Moscow (1991).