LITERATURE CITED

1. A. A. Gubaidullin, A. I. Ivandaev, and R. I. Nigmatulin, "Nonstationary waves in a liquid containing gas bubbles," Dokl. Akad. Nauk SSSR, 226, No. 6, 1299-1302 (1976).
2. V. E. Nakoryakov, V. V. Sobolev, and I. R. Shreiber, "Long-wave perturbations in a gas—liquid mixture," Izv. Akad. Nauk SSSR, Mekh. Zhidk. Gaza, No. 5, 71-76 (1972).
3. V. E. Nakoryakov, "Hydrodynamics of two-phase flows," in: Hydrodynamics and Heat Exchange in Two-Phase Media [in Russian], ITF Sib. Otd. Akad. Nauk SSSR, Novosibirsk (1981), pp. 5-30.
4. D. Batterworse and G. Hewitt (eds.), Two-Phase Flow and Heat Transfer [Russian translation], Énergiya, Moscow (1980).
5. R. I. Nigmatulin, Foundations of Mechanics of Heterogeneous Media [in Russian], Nauka, Moscow (1978).
6. I. V. Vinogradova, O. V. Rudenko, and A. P. Sukhorukov, Wave Theory [in Russian], Nauka Moscow (1979).

INFLUENCE OF PHYSICAL AND SCHEMATIC VISCOSITY IN THE ANALYSIS OF THE NEAR WAKE BEHIND A DISC

I. A. Belov, S. A. Isaev,
M. I. Nisht, and A. G. Sudakov

UDC 532.517.4

The influence of the physical and schematic viscosity on the results of computations performed within the framework of viscous and ideal medium models is analyzed in an example of uniform incompressible fluid flow around a disc.

1. The development of the near wake behind a disc is a typical example of the flow around bodies with a fixed site of flow separation on their surface. Investigation of the influence of viscosity in computing such flows is of important value in the determination of integral and local characteristics of poorly streamlined bodies, particularly the base pressure. Interest is also stimulated in the computation of separation flows by the practical utilization of flow control principles because of the premeditated formation of developed circulation zones near the streamlined bodies (see [1], for example). Separation flows can be modelled correctly on the basis of the solution of the system of complete nonstationary Navier—Stokes equations. The complexity of realizing such an approach even in investigations of stationary separation flows for significant Reynolds numbers is well known. The initial system of time-averaged Navier—Stokes equations in Reynolds form is not closed in this case and requires reliance on semiempirical models of turbulence which describe such flow structural elements of different scale as thin shear layers and vortex formations, whose dimensions are commensurate with the size of the body being streamlined.

In a number of cases, particularly in the solution of problems about the flow around a body with sharp edges at ultimately high Reynolds numbers when the influence of molecular viscosity on the flow becomes insignificant, the difficulties in solving the complete Navier—Stokes or the Reynolds equations resulted in the development of methods to compute separation flows that are based on a model of an ideal medium [2]. Let us mention just two, the method of discrete vortices and the method of coarse particles, whose detailed description is given in [3] and [4], respectively. The satisfactory agreement between the computed results obtained by using these methods and the experimental data (on body drag, flow configuration, etc.) affords a basis for the assumption that such an approach is justified in the consideration of fully developed turbulent flows. Let us note that modelling the turbulence is here presumably associated with singularities in the numerical realization of the analysis that

Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 50, No. 3, pp. 390-396, March, 1986. Original article submitted February 1, 1985.

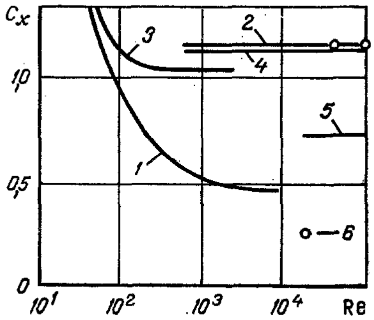

Fig. 1. Dependence of the frontal drag coefficient c_x on the Reynolds number Re. Solution of the Navier–Stokes equations (curves 1, 3) by using the Leonard scheme (1), and the hybrid scheme (3; data from [10]); solution of the Reynolds equations closed by using the $k - \varepsilon$ model of turbulence (2 and 4) by using the Leonard scheme (2) and the hybrid scheme (4); computation by the method of discrete vortices (5); 6) experimental results [6].

are due to discretization of the problem in both time and space, or as is customarily considered, in the influence of the artificial computational viscosity solution on the results.

Taking the above-mentioned into account, the numerical modelling of separation including turbulent flows assumes the necessity of distinguishing the action of the physical or molecular, computational of schematic and turbulent viscosity mechanisms if the latter is inserted into the consideration of the turbulence model. The purpose of the investigation is therefore an analysis of the influence of the kinds of viscosity mentioned on the solution of the problem of uniform incompressible fluid flow around a disc in the laminar and turbulent modes on the basis of comparing the results of a flow analysis obtained by different finite-difference methods and the method of discrete vortices with experimental data on local and integrated characteristics presented in [5, 6]. The velocity U_0 and the density ρ of the unperturbed flow as well as the disc radius R are selected as characteristic parameters. The disc is assumed infinitely thin in the computations.

2. General approaches to the schematization of separation flows and to the development of mathematical models of these flows are formulated in the monograph [3]. Their basic assumptions reduce to the following. The medium is considered ideal and incompressible. Flow separation is postulated on sharp edges and breaks in the lifting surface. Vortex surfaces are here shed from the edges and breaks and the velocities and rarefaction turn out to be finite here. To set up the nature of the ultimate separation flow, a nonstationary problem is solved in the general case, i.e., the whole process of flow formation is studied.

The mathematical formulation of the nonstationary problem of separation flow around a thin disc is described by the continuity equation in the form of the Laplace equation, boundary conditions on the disc surface, conditions on the vortex sheet and at infinite removal from the disc and its wake, the Chaplygin–Zhukovskii hypothesis on finiteness of the velocity at the sharp edge of the disc with which the vortex sheet converges, and also given initial conditions. Formation of the vortex wake with time is described by differential equations of the free vortex motion over liquid particle trajectories in an ideal medium. Since the medium is ideal, then the intensities of these vortices with time do not change, and only their position in space changes. The pressure on the disc and in the perturbed nonstationary flow are calculated by using the Cauchy–Lagrange integral.

The mentioned nonlinear model for separation flow around a thin disc by an ideal incompressible fluid is realized on an electronic computer by using the method of discrete vortices [3]. The disc and the vortex surface convergent with its edge are modeled by systems of discrete annular vortices (attached and free). The boundary condition about nonpenetration of the medium through the disc surface (it is satisfied at the finite number of checking lines), the condition about invariance of the circulation around a closed fluid contour in time, and the Chaplygin–Zhukovskii hypothesis are used to find their intensities at each computing time. Consequently, the boundary value problem reduces to solving a system of linear algebraic equations for the unknown attached and free vortex circulations at each computing time. The nonstationary problem is solved in steps by going from one computing time to another. Formation of the vortex structure in the wake is excecuted at the time under consideration by using the vortex circulations calculated in the preceding computing time.

The solution of the problem of separation flow around a disc on the basis of the method elucidated above is obtained when modeling the disc by ten discrete annular vortices, with a computational spacing in dimensionless time of $\Delta\tau = 0.1$ ($\tau = tU_0/R$ is the dimensionless time). Formation of the limit flow is realized for disc motion according to the law $U(\tau) = 0$, $\tau \leq 0$; $U(\tau) = U_0$, $\tau > 0$. The flow parameters, averaged with respect to time, in the wake

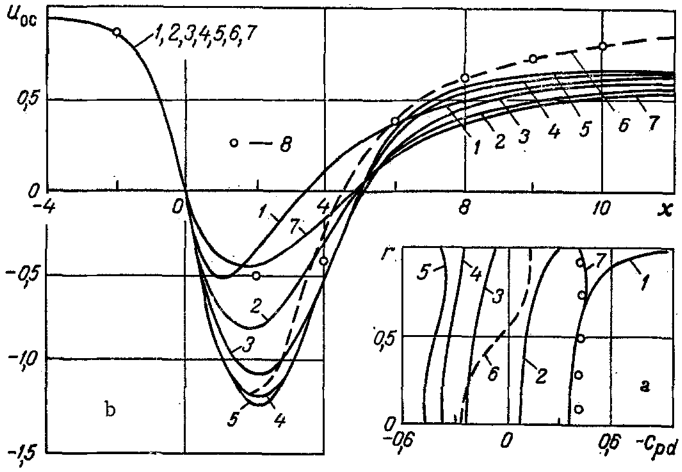

Fig. 2. Profiles of the base pressure coefficient on the rear surface of the disc (a), velocity distribution on the axis of symmetry (b). Solution of the Navier—Stokes equation by using the Leonard scheme: 1) Re = 40; 2) 10^2; 3) $2.5 \cdot 10^2$; 4) 10^3; 5) 10^4; 6) computation by the method of discrete vortices; 7) solution of the Reynolds equations closed by using the k — model of turbulence, by using the Leonard scheme for Re = $3.5 \cdot 10^4$; 8) experimental data [5].

and the disc drag were determined by the termination of the transient of flow formation at the time $\tau = 20$.

3. Modeling the stationary axisymmetric flow around a disc by a viscous incompressible fluid is based on solving the system of complete Navier—Stokes equations in the laminar mode by the method of control volume, and the system of complete Reynolds equations by using the semiempirical turbulence model k — ε in the turbulent mode. The initial system of motion equations in natural variables, supplemented in the turbulent flow case by differential equations for the turbulent energy fluctuations k and its dissipation rate ε is written is canonical divergent form in the cylindrical coordinates x, r, where x is the axial, and r the radial coordinate, respectively. Such a mode of writing permits constructuion of a conservative difference scheme that possesses the advantage that the conservation laws for any computation cell are satisfied to roundoff error accuracy. To solve the problem we introduce a computational mesh of "boomerang" type [7] with grid lines parallel to the coordinate directions and distributed with condensation near the disc surface and its sharp edge (minimal mesh spacing is 0.02).

Either a scheme with one-sided differences opposite to the flow or a "hybrid" scheme combining one-sided and central differences are used to assure stability of the computational procedure in representations of the convective terms of the transport equations (see [7], say). The error in solving the problem can turn out to be significant in real meshes limited by the electronic computer possibilities when modeling separation flows by using such schemes, especially at high Reynolds numbers. Its magnitude depends on the schematic viscosity due not only to the size of the computational cell but also to the magnitude of the local stream velocity as well as the streamline orientation relative to the mesh lines [7]. Zones with large stream parameter gradients, i.e., shear layers that develop on the separation domain boundary, are subjected to the greatest distortions. The influence of the schematic viscosity can be diminished substantially because of the increase in the order of approximation of the convective terms of the motion equations. Utilization of the scheme proposed by Leonard, with quadratic differences opposite to the stream [8], in this paper permits us to improve the quality of the modeling of the circulation flow in the near wake behind the disc and also to give a more correct estimate of the influence of turbulence effects on the flow than previously. It should be noted that the remaining terms in the equations are approximated by using central differences. Following [8], where it is shown that the solution for the turbulence characteristics k and ε are constructed by the "hybrid" scheme. The pressure correction method applied is substantially the analog of the splitting method at the finite-difference level. The difference equations obtained are solved by the method of linear scanning [8]. When modeling the turbulent flow around a disc, a $k - \varepsilon$ model with a set of universal semiempirical constants [8], approved for the computation of near-wall flows with developed turbulence is used.

The boundary conditions in the problem are set up in the usual manner: The unperturbed stream parameters are given at the input boundary, "soft" boundary conditions at the upper and output boundaries, and symmetry conditions on the axis of symmetry. The turbulence characteristics at the input boundary were selected from the condition of their minimal influence on the flow within the computational domain. The turbulent fluctuation energy ($k = 1.5 \cdot 10^{-4}$) is given by starting from the 1% level of unperturbed flow turbulence, while its dissipation rate is chosen so that the magnitude of the turbulent viscosity coefficient would be of the same order of magnitude as the molecular transport coefficient. Therefore, turbulence is generated in the stream because of the flow around the disc. The stream parameters and turbulence characteristics in the cells adjoining the walls are determined by the method of near-wall functions with the influence of the pressure gradient taken into account [9]. The adhesion condition is realized on the disc surface in the laminar mode. The distance between the disc and the left and right and upper boundaries of the computational domain are determined from the condition of no influence of posed boundary conditions on the stream parameter distribution in the neighborhood of the disc in numerical experiments, and are selected equal to 16, 24, and 18, respectively. Solution of the problem for the turbulent mode of the flow around a disc is obtained on a mesh with a 60 × 30 quantity of nodes on the average within 1500-2000 iteration steps with relaxation coefficients not exceeding 0.2-0.25, here the convergence of the solution was determined by setting up the turbulence characteristics in the computational field. The solution corresponding to the stationary laminar mode of the flow around a disc required on the order to 500-1000 iteration steps, where the convergence of the iteration process was determined by the degree of build-up of the stream parameters near the disc.

4. Computations of the viscous incompressible fluid flow around a disc on the basis of solution of the Navier—Stokes equations by using a Leonard difference-scheme of second-order approximation were performed in the $40\text{-}10^4$ range of Reynolds number variation. It should be noted that the numerical flow model has a physical analog and corresponds to the laminar mode of the flow around a disc only up to a certain critical Reynolds number Re_{cr} (on the order of 500) above which the flow becomes turbulent in the wake behind the disc. Consequently, the results obtained for $Re > Re_{cr}$ are methodological in nature and display the influence of convective transport processes on the flow to a definite degree for large Reynolds numbers. Comparing the results of computations corresponding to different Reynolds numbers, and also with the results of a computation executed by using the method of discrete vortices ($Re \to \infty$), permits estimation of the influence of the viscosity caused by molecular transport effects on the separation flow. The comparison of the results noticed above with the data of a numerical investigation of the flow around a disc by a hybrid scheme which is presented in [10], permits giving a representation of the role of the schematic viscosity in the formation of separation flows on the basis of difference schemes of first-order approximation. Computations of the turbulent mode for the flow around a disc at large Reynolds numbers (on the order of 10^4 and higher) were performed on the basis of solving the Reynolds equations with the inclusion of a semiempirical model of turbulence $k - \varepsilon$. Comparison of the numerical results obtained with the experimental data of an investigation of the flow around thin discs in wind tunnels [5, 6] permits estimation of the influence of turbulent effects on the separation flow, and also the estimation of the accuracy of modeling the flow under consideration on the basis of different numerical schemes.

Certain results of computations and available experimental data on the integral and local characteristics of the flow around a disc are represented in Figs. 1 and 2: The dependences of the frontal drag coefficient on the Reynolds number; the static pressure profiles on the rear surface of the disc, and the velocity distribution on the axis of symmetry. As follows from the computational results presented, the solution of the Navier—Stokes equation is asymptotic in nature as the Reynolds number increases. As is seen from Fig. 2a, the reason for the monotonic diminution in the frontal drag coefficient c_x is the rise in base pressure. It should be noted taht the maximal magnitude of the reverse flow velocity in the near wake behind the disc increases with the rise in Re, and exceeds the unperturbed stream velocity for $Re > 250$ (Fig. 2b). Here there is good agreement between the results of computing the flow around a disc by the method of discrete vortices and the almost asymptotic solution of the Navier—Stokes equations with small schematic viscosity. This is apparently a consequence of the overwhelming influence of convective processes, that are reproducible well by these models, on the flow.

Application of a second-order approximation scheme results in a significant diminution in the influence of schematic viscosity on the solution as compared with rougher computational schemes. The lack of a physical analog for the solutions of the Navier–Stokes equations (see Fig. 1) for small values of the physical and schematic viscosities (when using the Leonard scheme) indicates the dominant influence of turbulent transport processes in the real flow. Indeed, computations of the turbulent flow around a disc on the basis of using a $k - \varepsilon$ turbulence model yield results that are in good agreement with experimental data on the integral and local characteristics.

The results of a computation of the Navier–Stokes equations by a hybrid scheme are in quantitative agreement with experiment data starting with Re = 100-250, and diverge significantly from the results of a computation by a more exact Leonard scheme. This indicates the governing influence of the schematic viscosity in the formation of the circulation flow in the near wake behind the disc and in the similarity of the action of the schematic viscosity mechanisms in numerical modeling of the flow and the turbulent transport in its physical analog.

Therefore, comparing the results of numerical and physical modeling indicates that the flow around a disc is shaped under the action of convective and diffusion transport mechanisms. Where the diffusion process can be reproduced by not only the physical viscosity (both molecular and turbulent) in the numerical modeling of the flow but also because of the schematic viscosity introduced in discretization of the transport equations. Diminution of the schematic viscosity in combination with utilization of the semiempirical $k - \varepsilon$ model of turbulence based on the turbulent viscosity concept permits a sufficiently exact determination of not only the integral but also the local characteristics of the flow around a disc., i.e., raising the quality of modeling the separation flow as a whole.

NOTATION

x, r, Axial and radial coordinates (in fractions of the disc radius R); ρ, fluid density; μ, dynamic viscosity; p, pressure; u_{oc}, velocity component on the axis of symmetry (referred to the unperturbed stream velocity U_o); c_{pd}, base pressure coefficient ($c_p = 2p/(\rho U^2_o)$); k, energy of turbulent fluctuations (referred to U^2_o); ε, rate of turbulent energy dissipation (referred to $\nu U^2_o/R^2$); c_x, frontal drag coefficient (normalized with respect to the quantity $\pi\rho U^2_o R^2/2$); and Re, Reynolds number ($Re = \rho U_o R/\mu$).

LITERATURE CITED

1. I. A. Belov, Interaction between Nonuniform Streams and Obstacles [in Russian], Mashinostroenie, Leningrad (1983).
2. O. M. Belotserkovskii, S. M. Belotserkovskii, Yu. M. Davydov, and M. I. Nisht, Modeling Separation Flows on an Electronic Computer [in Russian], Scientific Council on the Complex Problem "Cybernetics," USSR Acad. Sci., Moscow (1984).
3. S. M. Belotserkovskii and M. I. Nisht, Separation and Separation-Free Flow around Thin Wings by an Ideal Fluid [in Russian], Nauka, Moscow (1978).
4. O. M. Belotserkovskii and Yu. M. Davydov, Method of Coarse Particles in Gasdynamics [in Russian], Nauka, Moscow (1982).
5. T. Carmody, "Wake development behind a disc," Trans. ASME, Theoret. Princip. Eng. Computat. [Russian translation], 86, No. 4, 281-296 (1964).
6. T. Morel and M. Bonn, "Flow around two circular discs located in tandem," Trans. ASME, Theor. Princip. Eng. Compuyat. [Russian translation], 102, No. 1, 225-234 (1980).
7. S. Patankar, Numerical Methods of Solving Heat Transfer and Fluid Dynamics Problems [in Russian], Énergoizdat, Moscow (1984).
8. M. A. Leshtsiner and W. Rody, "Computation of annular and twin parallel jets by means of different finite-difference schemes and turbulence models," Trans. ASME, Theor. Princip. Eng. Computat. [Russian translation], 103, No. 2, 299-308 (1981).
9. E. E. Khalil, "Numerical computations of turbulent reacting combustor flows," Numerical Methods in Heat Transfer, R. W. Lewis, K. Morgan, and O. C. Zinekiewicz (eds.), Wiley (1981), pp. 489-510.
10. I. A. Belov and N. A. Kudryavtsev, "Computation of laminar incompressible fluid flow around a disc and cylinder," Inzh.-Fiz. Zh., 42, No. 2, 290-295 (1982).