NUMERICAL INVESTIGATION OF THE EFFECT OF VISCOSITY ON SEPARATED FLOW PAST AN AUTOMOBILE PROFILE IN THE PRESENCE OF A MOBILE SCREEN

S. A. Isaev

The numerical simulation of laminar and turbulent separated flows of an incompressible viscous fluid past a two-dimensional complex-shaped body has been made on the basis of a factorized computational algorithm and H-like orthogonal grids in the presence of a mobile screen within a wide range of viscosity variation.

1. In the last few decades, the dynamics of large-scale vortex structures in variation of the viscosity of liquid media has invariably provoked interest among aerohydraulicmechanical engineers. To a large extent this is associated with the progress in numerical simulation of separation flows, the development of admissible computational algorithms, and the growth in the computational resources of computers. It should be noted that only at the end of 1970s–beginning of 1980s (see, e.g., [1]) were the requirements imposed on selecting the schemes of an elevated (not less than second) order of approximation substantiated to achieve appropriate accuracy of the solution of the Navier–Stokes equations in predicting the characteristics of complex flows with circulation zones.

It is of interest to trace the basic tendencies in numerical studies of the evolution of the structure of stationary separated flows as the Reynolds number increases. The problems considered are certainly of a fundamental character and are reflected, in particular, in L. G. Loitsyanskii’s well-known monograph [2].

It is historically established that two types of flows with circulation zones – closed flow in a square cavity with a moving boundary and flow of a homogeneous incompressible fluid past a transverse cylinder – has been subject to an extremely detailed analysis (see reviews in [3, 4]). However, as has been noted in [3], it is expedient to classify the conducted studies by the types of separation. Thus, it becomes possible to distinguish and unite into one group the works dealing with the consideration of flow past a disk [5], a cylinder, their combination [6], a stepped cylinder [7], and flow in a channel with a backward step [8]. The mentioned problems of a cavity and a cylinder should be referred to another group, and the problems of flow past a sphere and a profile should be added to them [see, e.g., [9]].

The effect of physical viscosity on the structure of the separated flow in the indicated problems manifests itself, as a rule, in a typical manner. As the Reynolds number increases, reciprocal-circulation flow becomes enhanced, and an increase in the sizes of primary large-scale vortices and the origination and evolution of the system of secondary vortices are observed. The presence of the walls bounding the flow exerts a substantial effect on the vortex dynamics.

This work is devoted to analysis of a little-studied flow near a profile of complex geometry near a mobile plane screen. Certain features of a turbulent mode of flow past such a profile were considered in [10-13]; however, the effect of viscosity was not studied in detail. The present work makes up for this deficiency.

2. The used methodology of numerical simulation of two-dimensional uniform flow of an incompressible viscous fluid past a body of arbitrary geometry is based on solution, within the framework of splitting by physical processes, of the initial system of Navier–Stokes equations for a laminar flow mode and the system of Reynolds equations for a turbulent flow mode, which is closed by a two-parameter dissipation model of
Fig. 1. Fragments of the \(H\)-type orthogonal computational grid around the profile of complex geometry near the mobile screen (the entire calculation region (a) and its central part (b)) and the dependences of the coefficients \(C_x(1), C_{xp}(2), C_{sd}(3), C_{cf}(4),\) and \(C_{sm}(5)\) of the profile on the Reynolds number (c); experimental data 6-8 are taken from [12, 13].

The flow past a body in the presence of a mobile screen is considered in a stationary statement with allowance for flow separation within a wide range of variation of Reynolds numbers. The turbulent character of the flow with developed circulation zones at high Reynolds numbers is simulated within the framework of a modified phenomenological approach which is related to the introduction of eddy viscosity and allows for the effect of the curvature of streamlines on the characteristics of turbulence after the idea of Leschziner and Rodi. The use of a high-Reynolds version of the modified model is combined with the use of the method of wall functions developed by Launder and Spalding.

A system of governing equations in divergent form is written in curvilinear coordinates, matched with the contour of a streamlined body, for increments in the dependent variables: Cartesian components of velocity, pressure, and characteristics of turbulence (kinetic energy of turbulent fluctuations and dissipation rate of turbulent energy). Discretization of the governing differential equations is made by the finite-volume method on a curvilinear orthogonal grid of the \(H\)-type generated on the basis of an original elliptical procedure [11].

The computational model, which is based on the adoption of the idea of splitting by physical processes and realized in the SIMPLEC procedure for a pressure correction, is subdivided into a number of computational blocks. At the "predictor" stage, preliminary components of the velocity for "frozen" pressure fields and characteristics of turbulence are determined, and at the "corrector" stage, pressure is corrected on the basis of the solution of the continuity equation with subsequent corrections of the velocity field and calculation of the characteristics of turbulence and eddy viscosity.

The selection of a centered pattern with fixation of dependent variables to the center of the computational cell allows one to substantially simplify the computational algorithm and reduce the number of computational operations. The monotization of the pressure field, which is necessary in this approach, is made within the framework of the Rhee–Chou approach [3, 10].

A high stability of the computational procedure is ensured by using, for discretization purposes, convective terms of the equations in the implicit side of one-side counter-flow differences, damping nonphysical oscillations through the introduction of artificial diffusion, and using pseudo-time stabilizing terms. The calculating efficiency of the computational algorithm increases when use is made of the method of incomplete matrix factorization for solving systems of nonlinear algebraic equations. Rather high accuracy of calculation is determined by discretization of the explicit side of the equations by the scheme of the second-order approximation, including discretization of the convective terms of the equations by Leonard's quadratic counter-flow scheme. This methodology allows minimization of the effects of "numerical" diffusion, which are especially substantial in calculations of separated flows.

3. Figures 1-4 sum up some results of calculation of laminar and turbulent stationary flows past a profile of complex geometry, representing a two-dimensional model of a Volkswagen automobile, in the vicinity of a mobile screen with variation in the Reynolds number within the range from \(10^2\) to \(10^7\). To solve the