**Numerical study of flow in a plane suddenly expanding channel based on Wilcox and two-fluid turbulence models**

**Z M Malikov**, **F Kh Nazarov**, **Z J Omonov** and **S K Abdukhamodov**

1 Akfa University, Tashkent, Uzbekistan
2 Tashkent state technical university, Tashkent, Uzbekistan

malikov.z62@mail.ru, farruxnazar@mail.ru, zarshidomonov96@gmail.com, Sardor.abdukhamdov.z62@mail.ru

**Abstract.** The article presents the results of a numerical study of the flow structure in a flat channel in the zone of its sudden expansion. The calculations are based on the two-fluid turbulence model and the Wilcox turbulence model. The stationary solution of the problem was obtained by the establishment method, for which non-stationary equations of hydrodynamics were used. The paper investigates the velocity fields and turbulent stresses of the flow, as well as the drag coefficient in different sections of the channel. The main calculations were performed on a uniform grid with 300 × 90 nodes. For the difference approximation of the original equations, the control volume method of the second order in space is used. The correctness of the results was confirmed for Reynolds numbers Re = 36000 by comparison with experimental data found in the literature.

**Key words:** Navier – Stokes equations, flat channel with a backward-facing step, separated flow, two-fluid model, control volume method, turbulent stresses.

**1. Introduction**

The study of separated flows has a great importance from both fundamental and applied points of view. A classic example for separated flows is a channel flow with sudden expansion. Fragments of fluid flow in suddenly expanding channels are found in various technical devices and structures. A sharp change in the geometry of the channel wall or the surface of the streamlined body can lead to separation of the flow and significantly change its kinematic structure. An example of such a flow is the movement of an incompressible viscous fluid in a flat channel with a backward-facing step, in which there are both flow separation at the edge of the step with its subsequent joining downstream to the lower wall of the channel, and the liquid recirculation zone immediately behind the step. Moreover, depending on the ratio of the flow inertia and viscous forces, there can be several such recirculation zones and their accompanying separation and flow reattachment points to the channel walls.

The first calculations of stationary two-dimensional laminar separated flows of an incompressible fluid in channels were obtained by Blasius in 1910 analytically in the form of series [1]. Later, this problem was used by many researchers to study the mechanisms of separated flows and to test difference schemes for solving the Navier-Stokes equations. Due to their great practical importance, such flows have been studied theoretically and experimentally for both laminar [2-4] and turbulent [5-7] regimes of motion of an incompressible and compressible fluid.
At present, it is obvious that when formulating the problems of calculating separated flows with vortex formations, it is necessary to use not the approximate equations of the boundary layer, but the complete Navier-Stokes equations. It is well known that the numerical solution of problems on the motion of a viscous incompressible fluid based on the Navier-Stokes equations is complicated not only by their nonlinearity, but also by the absence of an explicit equation for determining the pressure. In the work [8], a similar problem was solved, where a swirling flow undergoes a sudden expansion. For this, a two-fluid turbulence model was used. For the numerical implementation of discrete equations, the control volume method was used, and the pressure correction at the velocity was carried out by S.V. Patankar's method.

The paper considers the flow of a turbulent flow. It is known that currently there are basically three approaches to the problem of turbulence. The first approach uses Direct Numerical Simulation (DNS) methods [9, 10], while the second uses Large Eddy Simulation (LES) [11]. The third approach uses the Reynolds-Averaged Navier-Stokes Equations (RANS) method. This system of equations is not closed and various models are used to close it, which are called RANS turbulence models. Today there are over 100 different RANS turbulence models in the world. The NASA database [12] provides a comparative analysis of various semi-empirical models.

The purpose of this work is to apply a two-fluid turbulent model to solve an internal fluid flow problem in a flat channel with a sudden one-sided expansion and compare the numerical results with experimental data from the NASA database [12], as well as compare with the results of the well-known Wilcox model.

2. Physical and mathematical formulation of the problem

A two-dimensional turbulent flow in a flat channel with a sudden expansion in the form of a step is considered. The physical picture of the analyzed flow and the configuration of the computational domain are shown in Fig. 1. This problem is experimentally well studied and entered into the NASA database [12].

![Figure 1. Scheme of the computational domain in a flat channel with a backward-facing step](image)

To study the steady motion of a fluid, a two-fluid turbulence model and a Wilcox model were used. To obtain a stationary solution to the problem posed, the establishment method was used, for which a non-stationary problem was considered, the solution of which tends to a stationary solution for a sufficiently long time. The two-fluid model is described in detail in [8], as well as in general terms in [13, 14]. In these works, it was shown that the system of equations for a two-fluid model of a turbulent incompressible flow in a Cartesian coordinate system has the following form:
\[
\begin{align*}
\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} &= 0, \\
\frac{\partial U}{\partial t} + U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} + \frac{\partial p}{\partial x} &= \frac{1}{\text{Re}} \left( \frac{\partial^2 U}{\partial y^2} + \frac{\partial^2 U}{\partial x^2} \right) - \frac{\partial v u}{\partial y} - \frac{\partial u u}{\partial x}, \\
\frac{\partial V}{\partial t} + U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} + \frac{\partial p}{\partial y} &= \frac{1}{\text{Re}} \left( \frac{\partial^2 V}{\partial y^2} + \frac{\partial^2 V}{\partial x^2} \right) - \frac{\partial v v}{\partial y} - \frac{\partial u v}{\partial x}, \\
\frac{\partial u}{\partial t} + U \frac{\partial u}{\partial x} + V \frac{\partial u}{\partial y} &= -(1 - C_s) \frac{\partial U}{\partial y} v + \frac{\partial}{\partial x} \left( \frac{2 \nu'_{xxx} \frac{\partial u}{\partial x}}{3} \right) + \frac{\partial}{\partial y} \left( \nu'_{yy} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right) - C_s u, \\
\frac{\partial v}{\partial t} + U \frac{\partial v}{\partial x} + V \frac{\partial v}{\partial y} &= C_s \frac{\partial U}{\partial y} u + \frac{\partial}{\partial x} \left( \nu'_{xy} \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right) + \frac{\partial}{\partial y} \left( \nu'_{yy} \left( \frac{\partial v}{\partial y} + \frac{\partial v}{\partial x} \right) \right) - C_v v.
\end{align*}
\]

Here,
\[
\nu'_{xx} = 3/\text{Re} + 2 \frac{u u}{S}, \nu'_{yy} = 3/\text{Re} + 2 \frac{u v}{S}, \nu'_{yy} = 3/\text{Re} + 2 \frac{v v}{S},
\]
\[
C_s = 0.2, \quad C_v = C_1 \lambda_{max} + C_2 \left[ \frac{|V|}{d} \right]^2, \quad S = |\partial U / \partial y - \partial V / \partial x|.
\]

In the above equations \(U, V\) the axial and radial components of the air flow velocity vector, respectively, \(p\) is the hydrostatic pressure, \(u, v\) are the relative velocities, \(\nu_t\) — the turbulent viscosity of the liquid, \(\nu'_{xx}, \nu'_{yy}, \nu'_{xy}\) are the effective molar viscosities, \(d\) is the closest distance to the solid wall.

The system of equations of the Wilcox turbulence model has the following form:
\[
\begin{align*}
\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} &= 0, \\
\frac{\partial U}{\partial t} + U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} + \frac{\partial p}{\partial x} &= \frac{1}{\text{Re}} \left( \frac{\partial^2 U}{\partial y^2} + \frac{\partial^2 U}{\partial x^2} \right), \\
\frac{\partial V}{\partial t} + U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} + \frac{\partial p}{\partial y} &= \frac{1}{\text{Re}} \left( \frac{\partial^2 V}{\partial y^2} + \frac{\partial^2 V}{\partial x^2} \right), \\
\frac{\partial k}{\partial t} + U \frac{\partial k}{\partial x} + V \frac{\partial k}{\partial y} &= P - \beta^* \frac{\partial}{\partial y} \left[ \nu + \sigma_k \frac{k}{\omega} \frac{\partial k}{\partial y} \right] \\
\frac{\partial \omega}{\partial t} + U \frac{\partial \omega}{\partial x} + V \frac{\partial \omega}{\partial y} &= \frac{\gamma \omega}{k} P - \omega^2 + \frac{\partial}{\partial y} \left[ \nu + \sigma_\omega \frac{k}{\omega} \frac{\partial \omega}{\partial y} \right].
\end{align*}
\]

Here
\[
P = \tau_{ij} \frac{\partial u_i}{\partial x_j}, \quad \tau_{ij} = \nu_i \left( 2S_{ij} - 2 \frac{\partial u_i}{\partial x_j} \right), \quad 2k \delta_{ij}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad \nu_i = \frac{k}{\omega}, \quad \omega = \max \left[ \omega, C_{lim} \sqrt{\frac{2S_{ij} S_{ij}}{\beta^*}} \right], \quad S_{ij} = \frac{1}{3} \frac{\partial u_k}{\partial x_i} \delta_{ij}.
\]

The constant coefficients of the Wilcox model are
\[ C_1 = 0.7825, \quad C_2 = 0.306, \quad \sigma_z = 1.0, \quad \sigma_w = 1.3, \quad \beta^* = 0.09, \quad \gamma = \frac{13}{25}, \quad C_{\text{lim}} = \frac{7}{8}. \]

The Poiseuille flow in the entire computational domain was specified as the initial conditions. For numerical calculations, an experimental velocity profile corresponding to the location of the entrance to the computational domain was set at the input. The adhesion conditions were set on the walls. At the output, the conditions for extrapolation of the second order of accuracy were set. The Reynolds number was in agreement with the experimental data [12].

3. Solution method

Numerical solutions of the presented systems of equations were carried out in physical variables velocity - pressure by physical splitting of the velocity and pressure fields. In this case, a checkerboard difference grid was used for the transfer equations by the control volume method. According to this method, the first stage of integration of equations for the two-fluid model will have the form:

\[
\left\{ \begin{array}{l}
\frac{\tilde{U}_{i,j} - U_{i,j}}{\Delta t} + U_{i,j} \frac{\partial U_{i,j}}{\partial x} + V_{i,j} \frac{\partial U_{i,j}}{\partial y} + \frac{\partial p_{i,j}}{\partial x} = \frac{1}{\text{Re}} \left( \frac{\partial^2 \tilde{U}_{i,j}}{\partial y^2} + \frac{\partial^2 U_{i,j}}{\partial x^2} \right) - \frac{\partial v^2}{\partial y} - \frac{\partial u u}{\partial x}, \\
\frac{\tilde{V}_{i,j} - V_{i,j}}{\Delta t} + U_{i,j} \frac{\partial V_{i,j}}{\partial x} + V_{i,j} \frac{\partial V_{i,j}}{\partial y} + \frac{\partial p_{i,j}}{\partial y} = \frac{1}{\text{Re}} \left( \frac{\partial^2 \tilde{V}_{i,j}}{\partial y^2} + \frac{\partial^2 V_{i,j}}{\partial x^2} \right) - \frac{\partial v v}{\partial y} - \frac{\partial u v}{\partial x}, \\
u_{i,j}^{n+1} - u_{i,j}^n + U_{i,j} \frac{\partial u_{i,j}^n}{\partial x} + V_{i,j} \frac{\partial u_{i,j}^n}{\partial y} = (1 - C_s) \left( \frac{\partial U_{i,j}^{n+1}}{\partial y} - v_{i,j} \right) + \\
+ \frac{\partial}{\partial x} \left( 2 v_{iy} \frac{\partial u_{i,j}^{n+1}}{\partial x} \right) + \frac{\partial}{\partial y} \left( v_{iy} \frac{\partial u_{i,j}^{n+1}}{\partial y} \right) - C_f u_{i,j}^{n+1}, \\
\frac{V_{i,j}^{n+1} - v_{i,j}^n}{\Delta t} + U_{i,j} \frac{\partial v_{i,j}^n}{\partial x} + V_{i,j} \frac{\partial v_{i,j}^n}{\partial y} = C_s \left( \frac{\partial U_{i,j}^{n+1}}{\partial y} - u_{i,j} \right) + \\
+ \frac{\partial}{\partial x} \left( 2 v_{iy} \frac{\partial v_{i,j}^{n+1}}{\partial x} \right) + \frac{\partial}{\partial y} \left( v_{iy} \frac{\partial v_{i,j}^{n+1}}{\partial y} \right) - C_f v_{i,j}^{n+1}.
\end{array} \right.
\]

For Wilcox's model:

\[
\left\{ \begin{array}{l}
\frac{\tilde{U}_{i,j} - U_{i,j}}{\Delta t} + U_{i,j} \frac{\partial U_{i,j}}{\partial x} + V_{i,j} \frac{\partial U_{i,j}}{\partial y} + \frac{\partial p_{i,j}}{\partial x} = \frac{1}{\text{Re}} \left( \frac{\partial^2 \tilde{U}_{i,j}}{\partial y^2} + \frac{\partial^2 U_{i,j}}{\partial x^2} \right) + \frac{v_{iy}}{\partial y}, \\
\frac{\tilde{V}_{i,j} - V_{i,j}}{\Delta t} + U_{i,j} \frac{\partial V_{i,j}}{\partial x} + V_{i,j} \frac{\partial V_{i,j}}{\partial y} + \frac{\partial p_{i,j}}{\partial y} = \frac{1}{\text{Re}} \left( \frac{\partial^2 \tilde{V}_{i,j}}{\partial y^2} + \frac{\partial^2 V_{i,j}}{\partial x^2} \right) + \frac{v_{ix}}{\partial x}, \\
\frac{k_{i,j}^{n+1} - k_{i,j}^n}{\Delta t} + U_{i,j} \frac{\partial k_{i,j}^n}{\partial x} + V_{i,j} \frac{\partial k_{i,j}^n}{\partial y} = P - \beta \sigma \frac{\partial k_{i,j}^{n+1}}{\partial y} + \frac{\partial}{\partial y} \left( \frac{v + \sigma \frac{\partial k_{i,j}^{n+1}}{\partial y}}{\partial y} \right), \\
\frac{\omega_{i,j}^{n+1} - \omega_{i,j}^n}{\Delta t} + U_{i,j} \frac{\partial \omega_{i,j}^n}{\partial x} + V_{i,j} \frac{\partial \omega_{i,j}^n}{\partial y} = \frac{\gamma \omega}{k} \frac{\partial \omega_{i,j}^{n+1}}{\partial y} + \frac{\partial}{\partial y} \left( \frac{v + \sigma \frac{\partial \omega_{i,j}^{n+1}}{\partial y}}{\partial y} \right).
\end{array} \right.
\]
It is easy to show that the design model has an order of accuracy $O(\Delta x, \Delta y, \Delta t)$. After fulfilling this condition, the pressure and velocities were found as follows:

$$V_{j+1} = \tilde{V}_j - \Delta t \frac{\Delta P_j}{\Delta y},$$
$$P_{j+1} = P_j + P_{\text{correction}}.$$

For the numerical solution of equation (7), a semi-implicit scheme was used, which is effectively implemented by the sweep method. Here $k$ is the iteration number. For each time layer, iteration continued until the condition

$$\left| \frac{\Delta P_i^{n+1} - \Delta P_i^n}{\Delta y} \right| < 10^{-6}$$

is fulfilled. The upstream circuit has the following form:

$$\frac{\Delta P_j^{n+1} - \Delta P_j^n}{\Delta y} \approx \frac{1}{\Delta y} \left( \tilde{U}_j - \tilde{U}_{j-1}, \frac{\Delta x}{\Delta x} \right).$$

(8)

For the numerical solution of the transport equation of system (3) and (4), a finite-difference scheme was used, which is exact value. The upstream circuit has the following form:

$$\frac{\Delta P_j^{n+1} - \Delta P_j^n}{\Delta y} = \frac{\Delta P_j^{n+1} - 2\Delta P_j^{n} + \Delta P_j^{n-1}}{\Delta y}.$$ 

(7)

where $\Delta P_j$ is an iteration parameter. When solving equation (7) for the time step $t$, it is possible to write down the value of the constant $a$, less than one and is selected from the condition of the rapid convergence of the numerical process. The Neumann condition is used as the boundary condition for the dependence of the pressure correction $P_{\text{correction}}$. According to the central difference scheme, the following equation is obtained:

$$\Delta P_j^{n+1} = \frac{\Delta P_j^{n+1} - 2\Delta P_j^{n} + \Delta P_j^{n-1}}{\Delta y} = \frac{\Delta P_j^{n+1} - 2\Delta P_j^{n} + \Delta P_j^{n-1}}{\Delta y}.$$ 

(6)

Equation (6) was solved by the iteration method, for which the following equations were used:

$$\Delta P_j^{n+1} = \frac{\Delta P_j^{n+1} - 2\Delta P_j^{n} + \Delta P_j^{n-1}}{\Delta y}.$$ 

(5)

Equations (3) and (4) the superscript "$\tilde{v}$" denotes the intermediate grid function for the velocity vector $\tilde{v} = \frac{\partial P}{\partial x}$, correction to pressure. After substituting to the velocities $\tilde{v}$ into the continuity equation, we obtain the Poisson equation for determining the correction to the pressure.
4. Results and discussions

In figure 2 shows graphs comparing the calculated and experimental data. The figures show the profiles of the longitudinal velocity U in various measured sections at distances from the entrance to the wide channel.

![Profiles of longitudinal U velocity in sections](image)

**Figure 2.** Profiles of longitudinal U velocity in sections x = 6H (a), x = 11H (b), x = 14H (c), x = 16H (d)

It can be seen from this figure that the results of the models generally coincide with each other and are in satisfactory agreement with the experimental data. Only at large distances is the agreement between the results of the two-fluid model slightly better.

In figure 3 shows the profiles of turbulent stress in various measured cross sections.
If the results of the two models for the velocity profile are in good agreement with each other, then the analysis of the results presented for the turbulent stress in figure 3 shows that the two-fluid model gives better results than the Wilcox model. According to the authors, this is due to the fact that the Wilcox model is based on the assumption of isotropic turbulence. It is known that turbulence in circulation flows becomes anisotropic. As for the two-fluid model, as shown in [8,13,14], it is capable of describing anisotropic turbulence.

In figure 4 shows the distribution of the coefficient of friction along the length of the channel.
Figure 4 shows that the results of the models are in good agreement with the experimental data.

5. Conclusions
The paper presents the numerical solutions of the two-fluid turbulence model and the Wilcox model as applied to the problem of the flow of an incompressible viscous fluid in a short flat channel with a backward-facing step. Changes in velocity, turbulent stress, and coefficient of friction are demonstrated. The results obtained, despite the use of a rather coarse homogeneous computational grid, are in good agreement with each other and with the experimental data. The numerical implementation of the models showed that the stability of the two-fluid model is significantly greater than that of the Wilcox model. For example, in the Wilcox model, the instability of the model occurred at, and the two-fluid model retained its stability even at. Therefore, the two-fluid model can be recommended for numerical studies of separated flows in practical problems.

6. References
[1] Horowitz G T and Maldacena J 2004 The black hole final state J. High Energy Phys. JHEP 02 (2004) 008
[2] Honji H 1975 The starting flow down a step J. Fluid Mech. 69, Pt. 2 P 229–240
[3] Sinha S P, Gupta A K, Oberay M M 1981 Laminarnoye otryvnoye obtekaniye ustupov i cavern. Pt 1 Tetcheniye za ustupom Raketnaya tekhnika i kosmonavtika vol 19, No 12 P 33–37
[4] Armaly V F, Durst F, Pereira J S F, Schonung V 1983 Experimental and theoretical investigation of backward-facing step flow. J. Fluid Mech. 127 P 473–496
[5] Chjen P Otryvnyye techniya 1972-1973 [Separated currents] (M: Mir) vol 1 300 p, vol 2 280 p, vol 3 354 p
[6] Gogish L V, Stepanov G Yu 1979 Turbulentnyye otryvnyye techniya [Turbulent separated flows] (M: Nauka) 368 p
[7] Le P, Moin P, Kim J 1997 Direct numerical simulation of turbulent flow a backward-facing stap J. Fluid Mech. 330 P 349–374
[8] Malikov Z 2020 “Mathematical Model of Turbulence Based on the Dynamics of Two Fluids”. Applied Mathematical Modeling vol 82 P 409–436
[9] Versteegh T A, Nieuwstadt F T 1997 Turbulent Budgets of Natural Convection in an Infinite, Differentially Heated, Vertical Channel Intern. J. Heat Fluid Flow. vol 19 P 135

[10] Boudjemadi R, Maupu V, Laurence D, Le Quere P 1996 Direct Numerical Simulation of Natural Convection in a Vertical Channel: A Tool for Second-Moment Closure Modelling Proc. Engineering Turbulence Modelling and Experiments 3 (Amsterdam: Elsevier) P 39

[11] Peng S-H, Davidson L 2001 Large Eddy Simulation of Turbulent Buoyant Flow in a Confined Cavity Intern. J. Heat Fluid Flow. vol 22 P 323

[12] Cristopher R Responsible NASA official. “Turbulence modeling Resource. NASA Langley Research Center”, http://turbmodels.larc.nasa.gov. (data obrashcheniya 04.04.2019)

[13] Malikov Z M, Madaliev M E 2020 Chislennoye modelirovaniye dvukh faznogo potoka v tsentrobezhnom separatore. Prikladnaya matematika i mekhanika (Moskva) [Numerical simulation of a two-phase flow in a centrifugal separator. Applied Mathematics and Mechanics] (Moskva) vol 84 No 5 P 590–611

[14] Malikov Z M, Madaliev M E 2020 Numerical Simulation of Two-Phase Flow in a Centrifugal Separator ISSN 0015-4628 Fluid Dynamics vol 55 No 8 pp 1012–1028