Numerical modeling of turbulent flows near axially symmetric surfaces

M M Gorokhov¹,², A V Korepanov¹,³, V A Tenenev¹ and S V Vologdin¹
¹ Kalashnikov Izhevsk State Technical University, 7, Studencheskaya street, Izhevsk, 426069, Russia
² Research Institute of Federal Penitentiary Service, 15a, Narvskaya street, Moscow, 125130, Russia
³ E-mail: kor-and@yandex.ru

Abstract. The relevance of performing numerical modeling of turbulent flows around axially symmetric surfaces is presented. A review of similar scientific works is carried out. A mathematical model of a turbulent axially symmetric flow around particles of an arbitrary shape surface is constructed. A numerical method for performing calculations is described. Numerical methods for constructing a curvilinear orthogonal grid have been developed. Examples of flow around particles are presented for the turbulent boundary layer.

1. Introduction
The study on the flow around axially symmetric bodies for the subcritical range of Reynolds numbers, from 10 to $2 \cdot 10^5$, was carried out in [1]. In [2], the mathematical model proposed in [1] was used to study the combustion of aluminum particles in a flow. The specified interval of Reynolds numbers is characterized by the fact that up to the separation point, the boundary layer on the body is laminar [3] and turbulence can be ignored in the calculations.

Similar calculations were carried out in the works of V.A. Gushchin [4], however, all calculations were carried out for subcritical Reynolds number.

In the current paper, a mathematical model is presented and numerically implemented, which makes it possible to carry out research at supercritical Reynolds numbers exceeding the value $2 \cdot 10^5$ where the boundary layer on the surface of a streamlined body is characterized as turbulent.

2. Mathematical model
In accordance with [1, 2], introduce an orthogonal curvilinear axisymmetric coordinate system $\xi, \eta$ (figure 1).

The equations describing a stationary viscous isothermal incompressible gas flow around an axially symmetric body, written in a curvilinear coordinate system, have the form:

$$F_\xi + G_\eta = P + R_\xi + H_\eta + S,$$  \hspace{1cm} (1)

where
Here $\rho$ – gas density; $p$ – pressure; $u$, $v$ – flow velocity components along the axes $x$, $y$; $U$, $V$ – contravariant velocity components along the axes $\xi$, $\eta$; $D$ – the Jacobian transformation of the original coordinate system $x$, $y$ into an orthogonal curvilinear system $\xi$, $\eta$; $\mu = \mu_0 + \mu_T$ – sum of molecular and turbulence eddy viscosity.

To take into account the turbulent flow regime in the boundary layer, it is necessary to use the turbulence eddy viscosity model. Numerical calculations of the flow parameters around the sphere showed that the application of the algebraic model of turbulence eddy viscosity [3] did not lead to the elimination of the early separation of the boundary layer and did not give the effect of a sharp decrease in the impedance coefficient. The one-parameter model of A.N. Sekundov was also considered [5]. The results were also unsatisfactory [6]. Next, a two-parameter model of turbulence eddy viscosity is considered [7].

Thus, to describe turbulent flow regimes, the equations for the transfer of turbulence energy and dissipation rate $\varepsilon$ should be added to the system of equations [7]:

$$
F_\xi + G_\eta = R_\xi + H_\eta + S,
$$

where

$$
F = yDU \begin{bmatrix} \rho E \\ \rho \varepsilon \end{bmatrix}, \quad
G = yDV \begin{bmatrix} \rho E \\ \rho \varepsilon \end{bmatrix}, \quad
R = y\mu \begin{bmatrix} E_\xi \\ E_\eta \end{bmatrix}, \quad
H = y\mu \begin{bmatrix} E_\xi \\ E_\eta \end{bmatrix}, \quad
S = yD \begin{bmatrix} \mu_T \frac{Q - \rho \varepsilon}{E} \\ c_\mu c_Q \frac{Q - c_2 \rho \varepsilon}{E} \end{bmatrix},
$$

$$
c_1 = 1.44, \quad c_2 = 1.92, \quad c_\mu = 0.09, \quad \mu_T = c_\mu \frac{E^2}{\varepsilon},$$

$$
Q = 2 \left( u_\xi \xi_x - u_\eta \eta_x \right)^2 + \left( v_\xi \xi_x + v_\eta \eta_x \right)^2 + \left( u_\xi \xi_x + u_\eta \eta_x + v_\xi \xi_x - v_\eta \eta_x \right)^2 + \left( v_\eta \xi_x \right)^2,
$$

$Q$ – turbulence generation.

The area of numerical integration (figure 1) is limited by the line of symmetry AF, ED, the body contour FE, input AB, output CD and external BC boundaries, which are located in the outer flow. The boundary conditions are as follows: on the body surface $U = 0$, $V = 0$. Taking into account the
logarithmic distribution of the average flow velocity, the boundary conditions for \( E \) and \( \varepsilon \) at the first node of the difference grid located outside the viscous sublayer are taken as follows:

\[
E_1 = \frac{u_*^2}{\sqrt{\frac{1}{c_\mu}}}, \quad \varepsilon_1 = \frac{u_*^3}{\chi y_1},
\]

where \( u_* \) – friction velocity determined by wall shear stress parameter \( \tau_0 \) \((\tau_0 = \sqrt{\mu u_*/\rho})\); \( \chi = 0.41 \) – the Karman constant.

Symmetry boundary conditions: \( U_\eta = V = E_\eta = \varepsilon_\eta = 0 \) ; at the input and upper boundaries \( U = D^{-1/2} u_\infty \), \( V = E_\xi = \varepsilon_\xi = E_\eta = \varepsilon_\eta = 0 \), \( u_\infty \) – the incoming flow velocity; at the outlet boundary \( U_\xi = V_\xi = E_\xi = \varepsilon_\xi = 0 \), \( p = p_\infty \), \( p_\infty \) – the undisturbed flow pressure.

3. Numerical method

For the numerical solution of the system of equations (1), the SIMPLE algorithm was used [8]. The algorithm is based on the derivation of finite-difference equations obtained using the control volume method and their numerical implementation on a shifted grid. The essence of such a grid is the calculation of pressure values and velocity components at various nodes.

Discrete analogs of the equations of motion of system (1) have the form:

\[
a_{i,j} u_{i,j} = \sum_k a_k u_k - \left[ yD(\xi_j p_{\xi_j} - \xi_j p_\eta)\right]_{i,j}, \quad (3)
\]

\[
a_{i,j} v_{i,j} = \sum_k a_k v_k - \left[ yD(\xi_j p_{\xi_j} + \xi_j p_\eta)\right]_{i,j} - \left( \frac{\mu v D}{y} \right)_{i,j}, \quad (4)
\]

where \( k \) – index, meaning the numbers of cells surrounding the cell under consideration \((i, j)\); the coefficients \( a_{i,j}, a_k \) are determined by analogy with [8].

Write equations (3, 4) in projections on the axis \( \xi, \eta \): multiplying equation (3) by \( \xi_j \), and equation (4) by \( \xi_j \) and adding them, a difference equation to determine \( U \) is obtained; multiplying equations (3) by \( -\xi_j \), and equation (4) by \( \xi_j \) and adding, their right and left sides, the difference equation to determine \( V \) is obtained. In accordance with the methodology for implementing the SIMPLE algorithm, denote the velocity field obtained using the approximate pressure field \( p^{*} \), though \( U^{*}, V^{*} \). For variables \( M^* = y D U^*, N = y D V^* \) the difference equations have the form:

\[
a_{M_{i,j}} M_{i,j}^* = \sum_k a_{kM} M_{i,j}^* + c_{M_{i,j}} \left( p_{i,j+1}^* - p_{i,j}^* \right) + b_{M_{i,j}}, \quad (5)
\]

\[
a_{N_{i,j}} N_{i,j}^* = \sum_k a_{kN} N_{i,j}^* + c_{N_{i,j}} \left( p_{i,j+1}^* - p_{i,j}^* \right) + b_{N_{i,j}}, \quad (6)
\]

where the coefficients \( a_M, a_N, c_M, c_N \) are obtained by integrating the equations of system (1) over the shifted control volumes for \( M \) and \( N \); \( i, j \) – over the shifted control volumes \( \xi, \eta \), the index \( k \) means the summation over the control volumes surrounding the considered volume, the indices \( M \) and \( N \) indicate the belonging to the corresponding variables.

In these equations, in addition to the source terms with pressure gradient components, sources appeared \( b_M, b_N \) which have the form:

\[
b_{M_{i,j}} = \sum_k a_{kM} \left[ M_{k}^{\xi} \left( y_{i,j} D_{k} - 1 \right) + v_{i,j} D_{k} \right] - \frac{\mu D^2 \xi_j \xi_j x \Delta \xi \Delta \eta N^{n}}{y} \right]_{i,j},
\]
\[ b_{Ni,j} = \sum_k \alpha_{nk} \left[ N_k^p \left( \frac{y_{lj}D_{lj}}{y_{lj}D_{lj}} - 1 \right) + \frac{y_{lj}D_{lj}}{y_{lj}D_{lj}} \left( N_k^p \varphi_k - M_k^p \psi_k \right) \right] \left( \frac{\eta D^2 \xi_j \Delta \xi \Delta \eta \Delta M^p}{y} \right)_{lj}, \]

where \( \varphi_k = D_k \left[ \xi_{ni,j} \xi_{nk} + \xi_{ni,j} \xi_{nk} \right] - 1, \psi_k = D_k \left[ \xi_{nj,i} \xi_{nk} - \xi_{ni,j} \xi_{nk} \right], \Delta \xi, \Delta \eta, \Delta M^p \) – sizes of control volumes in directions \( \xi, \eta \), \( \xi = \frac{x}{D}, \eta = \frac{y}{D} \) – metric coefficients; the index \( n \) means the number of the previous iteration.

In accordance with [7], the pressure is introduced in the form:

\[ p = p^* + p', \]  

where \( p' \) – pressure correction.

The equation for the pressure correction \( p' \) is derived by substituting into the continuity equation \( M^*_\xi + N^*_\eta = 0 \) the values of the variables \( M_{i,j} \) and \( N_{i,j} \) determined from the expressions:

\[ M_{i,j} = M^*_{i,j} + d_{Mi,j} \cdot (p_{i+1,j}^* - p_{i,j}^*), \]

\[ N_{i,j} = N^*_{i,j} + d_{Ni,j} \cdot (p_{i,j+1}^* - p_{i,j}^*), \]

where \( d_{Mi,j}, d_{Ni,j} \) determined according to [8].

Thus obtain:

\[ a_{Pi,j} \cdot p_{i,j}^* = \sum_k \alpha_{pk} \cdot p_{i,j}^* + b_{Pi,j}, \]

where \( b_{Pi,j} = (M^*_{i,j} - M^*_{i+j}) \cdot \Delta \eta_j + (N^*_{i,j} - N^*_{i,j+1}) \cdot \Delta \xi_i \).

A discrete analogue of equations (2) and has the form:

\[ a_{Phi,j} \cdot \Phi_{i,j} = \sum_k \alpha_{pk} \cdot \Phi_{i,j} + b_{Phi,j}, \]

where \( \Phi \) s a vector with the following components \( \Phi = [E, \varepsilon] \). The coefficients \( a_{Phi,j}, a_{Phi,j} \) are determined according to [9]; \( b_{Phi,j} \) contains a finite-difference approximation of \( \varepsilon \) equations (2).

The system of linear algebraic equations (5, 6, 11) was solved by the iterative Gauss-Seidel method in combination with the lower relaxation method.

Equation (7) was solved using the conjugate gradient method with regularization based on the incomplete Cholesky decomposition [9].

The numerical solution consists of a cyclically performed guess-correction procedure [8]:

- for an approximate value of pressure \( p^* \) given in the region of numerical integration, approximate values of velocities \( M^*, N^* \) are determined from equations (5, 6);
- from equation (7) the values of pressure correction \( p' \) are determined;
- from the relations (8, 9, 10), the actual values of pressure \( p \) and velocities \( M, N \) are determined;
- from equation (2) the values \( E \) and \( \varepsilon \) are determined;
- the residual value is calculated \( \vartheta = \left| p'/p_{\text{max}} \right| \) and if \( \vartheta > \zeta \) (\( \zeta = 10^{-4} \)), then t is necessary to return to step 1 using the obtained values \( U, V, p \) as approximate.

The iterative process ends if the condition \( \vartheta \leq \zeta \) is met.
The dependence $\vartheta$ on the number of iterations and the corresponding average identification on the surface of the value of the impedance sphere coefficient at $Re = 4.35 \cdot 10^5$ are shown in figure 2. It can be seen from the figure that the value $\zeta = 10^{-4}$ is sufficient, since on the interval of change $\zeta$ from $10^{-4}$ to $10^{-5}$ the $C_x$ value changes insignificantly (by less than $4\%$).

![Figure 2. Dynamics of establishing the mean integral surface impedance coefficient.](image)

4. Finite-difference grid

When calculating flows in arbitrary areas, either rectangular finite-difference grids with blocked areas or curvilinear grids adapted to the flow field are used.

The use of rectangular grids with locked areas makes the boundary conditions more difficult to implement. Such grids do not allow the refinement of the grid spacing near a solid surface, which is necessary when calculating flows with a high Reynolds number.

The complex shape of the surfaces of the investigated bodies requires the use of computational grids adapted to the flow conditions. Constructing curvilinear finite-difference grids for an arbitrary region is a rather difficult task. In addition, the property of grid orthogonality is essential for the SIMPLE algorithm [8]. However, the use of such computational grids greatly simplifies the calculation algorithm, formulation and numerical implementation of the boundary conditions, greatly facilitates the solution of the problem associated with the appearance of circuit viscosity, and makes it possible to thicken the grid lines near a solid surface.

To construct a curvilinear finite-difference mesh, two families of lines will be defined as function level lines $\xi(x, y), \eta(x, y)$, that satisfy the Laplace equations [9, 10]:

$$
\xi_{xx} + \xi_{yy} = 0, \quad \eta_{xx} + \eta_{yy} = 0,
$$

(12)

where $x$, $y$ – physical coordinates.

To find the coordinates of the nodes of the finite-difference grid (12) it is advisable to use equations for the functions $x(\xi, \eta), y(\xi, \eta)$:

$$
\alpha_1 x_{\xi\xi} + \alpha_2 x_{\xi\eta} + \alpha_3 x_{\eta\eta} = \beta_1,
\alpha_1 y_{\xi\xi} + \alpha_2 y_{\xi\eta} + \alpha_3 y_{\eta\eta} = \beta_2.
$$

(13)

where $\alpha_1$, $\alpha_2$, $\alpha_3$, $\beta_1$, $\beta_2$ – functions of $x_{\xi}$, $y_{\xi}$, $x_{\eta}$, $y_{\eta}$ [10].

To construct an orthogonal finite-difference mesh, it is necessary to put in equations (13) $\alpha_1 = \alpha_3 = 1$, $\alpha_2 = \beta_1 = \beta_2 = 0$ and require the Cauchy-Riemann equations to be satisfied on the boundary:

$$
x_{\xi} = y_{\eta}, \quad x_{\eta} = -y_{\xi},
$$

(14)

The task set will be carried out using the complex boundary element method [11].
The boundary $S$ of the region $\sigma$ (figure 3) is approximated by a broken line with the given coordinates of the nodes $z_k = x_k + i \cdot y_k$, $k = 1, 2, ..., L$ in the complex region $Z$. At the border sections $S$ the boundary values of one of the functions $\xi$ or $\eta$ are set: on $AB$ $\xi = 0$, on $BC$ $\eta = 1$, on $CD$ $\eta = (S - S_0)/(S_C - S_0)$, on $DA$ $\eta = 0$.

For an analytic function $\omega(z) = \xi + i \cdot \eta$ in a domain $\sigma$ with a boundary $S$, where $\xi$ and $\eta$ satisfy the equations (14), the Cauchy formula is valid

$$\omega(z) = \frac{1}{2 \cdot \pi \cdot i} \int_{\gamma} \frac{H(\gamma)}{\gamma - z} d\gamma, z \in \sigma, \gamma \in S,$$

(determining the values of the function at the interior points of the domain $\sigma$ from the known values $H(\gamma)$ of this function on the boundary, where

$$H(\gamma) = \omega_k \cdot \frac{\gamma - \gamma_k}{\gamma_{k+1} - \gamma_k} + \omega_{k+1} \cdot \frac{\gamma_{k+1} - \gamma_k}{\gamma_{k+1} - \gamma_k}, \gamma \in [\gamma_k, \gamma_{k+1}], k = 1, ..., L.$$}

Integration of equation (15) for each boundary node $z_k$ leads to a system of linear algebraic equations $L$ that determine the values of functions $\omega_k = \xi_k + i \cdot \eta_k$ at the nodes $z_k = x_k + i \cdot y_k$.

Using the obtained boundary values $\omega_k$, it is possible to write the Cauchy formula in the complex domain $\omega$, but for the function $z$:

$$z(\omega) = \frac{1}{2 \cdot \pi \cdot i} \int_{C} \frac{H(\gamma)}{\gamma - \omega} d\gamma, \omega = \Pi, \gamma \in C,$$

where $C$ – the border of the rectangle $\Pi$ in the plane $\xi$, $\eta$. Here the function $H(\gamma)$ takes the form:

$$H(\gamma) = z_k \cdot \frac{\gamma_{k+1} - \gamma_k}{\gamma_{k+1} - \gamma_k} + z_{k+1} \cdot \frac{\gamma_k - \gamma_{k+1}}{\gamma_{k+1} - \gamma_k}, \gamma \in [\gamma_k, \gamma_{k+1}], k = 1, ..., L.$$}

The integral of equation (16) is divided into $L$ integrals over the number of segments of the broken line of the boundary. For each given value $\omega_0 = \xi_m + i \cdot \eta_n$, where $\xi_m$, $\eta_n$ the given levels of functions $\xi(x, y)$, $\eta(x, y)$ ($m = 1, ..., M$, $n = 1, ..., N$) after integrating equation (16), taking into account equation (17) the values $z_0 = x_m + i \cdot y_m$ determining the coordinates of the grid nodes are found:

$$z_0 = \frac{1}{2 \cdot \pi \cdot i} \sum_{k=1}^{L} \left( \frac{\omega_0 - \omega_k}{\omega_{k+1} - \omega_k} - z_k \frac{\omega_k - \omega_{k+1}}{\omega_{k+1} - \omega_k} \right) \ln \left( \frac{\omega_{k+1} - \omega_0}{\omega_k - \omega_0} \right).$$

Since the calculations by the formula of equation (18) are associated with some error, then for strongly curved domains it is possible to obtain non-monotonic behavior of functions $x(\xi, \eta)$, $y(\xi, \eta)$.
and, therefore, an overlap of coordinate lines is possible. Therefore, to refine the position of the nodes at the final stage, it is advisable to solve the Laplace equations

\[ x_{\xi\xi} + x_{\eta\eta} = 0, \quad y_{\xi\xi} + y_{\eta\eta} = 0 \]

for given boundary conditions satisfying equation (14) and a given location of the boundary nodes, using the solution obtained above as the first approximation.

In the calculations, finite-difference grids were used with the following parameters: 150 grid nodes were set in the direction \( \xi \), 41 of them were located on the body surface, and 60 grid nodes were set in the direction \( \eta \). A further increase in the number of nodes in the directions \( \xi \) and \( \eta \) is impractical, since it leads to an insignificant change in the mean value of the surface impedance coefficient \( C_x \) (an increase in the number of nodes by a factor of two leads to a change in the value of \( C_x \) by \( \sim 0.1\% \)). The domain size in the direction \( \xi \) in front of the body changed from \( 3R \) to \( 15R \), behind the body from \( 15R \) to \( 30R \). The domains size in the direction \( \eta \) changed from \( 5R \) to \( 15R \), where \( R \) – the radius of the midsection of the body. All changes depend on the Reynolds number \( \text{Re} = \rho u_\infty^2 R / \mu \). With an increase in the Reynolds number, the size of the domain changed as follows: the size of the domain in front of the body and above the body decreased; the size of the area behind the body increased. The finite-difference grid was refined towards the body surface so that at least 5 nodes would fall into the laminar sublayer. An example of a finite difference grid is shown in figure 3.

![Figure 3. Example of a finite difference grid.](image)

5. Calculation results

The convergence of the numerical solution method was established by refining the difference mesh and varying the size of the numerical integration region. As a test problem, we considered the problem of flow around a sphere in the range of variation of the value of the Reynolds number from \( 10 \) to \( 10^6 \). Figure 4 shows the calculated values of the coefficient of the impedance of the sphere on the Reynolds number and the standard resistance curve approximated by the dependences [12, 13]:

\[
C_x = \frac{24}{\text{Re}} \left( 1 + 0.25 \sqrt{\text{Re}} + 0.0117 \text{Re} \right), \quad (1 \leq \text{Re} \leq 1000); \quad C_x = 0.44, \quad (1000 < \text{Re} \leq 2 \cdot 10^5). 
\]

![Figure 4. Ball drag coefficient versus Reynolds number.](image)

It can be seen from the figure that in the range of variation of the Reynolds number from \( 10 \) to \( 2 \cdot 10^5 \) the calculated and experimental values are in satisfactory agreement with the approximation dependences. At the Reynolds number \( \sim 3 \cdot 10^5 \) according to the experimental data, the laminar boundary
layer on the sphere turns into a turbulent one and the coefficient of turbulent viscosity is calculated from equations (2). In this case, calculations show that the value of the impedance coefficient decreases. This result is consistent with experimental data [3]. Comparison of the calculated and experimental values of $C_x$ in the range of Reynolds numbers from $3 \cdot 10^5$ to $10^6$ is presented in table 1.

**Table 1.** Comparison of calculated and experimental values of $C_x$.

| $Re$     | $C_x$ estimated | $C_x$ experiment [3] |
|----------|----------------|----------------------|
| 251300   | 0.297          | 0.313                |
| 298500   | 0.172          | 0.151                |
| 424500   | 0.209          | 0.143                |

Figure 5 shows the change in the flow field near the sphere for different values of the Reynolds number. As can be seen from the figures, the size of the circulation zone behind the sphere increases with an increase in the Reynolds number. When the critical value of the Reynolds number $\sim 3 \cdot 10^5$ is reached, the circulation zone is displaced downstream and its size decreases, which is confirmed by the experimental data [3, 14].

![Figure 5](image)

**Figure 5.** Influence of the Reynolds number on the flow pattern around a sphere.

The highest values of turbulent viscosity correspond to the circulation zone (figure 6).
Figure 6. Distribution of turbulent viscosity related to molecular viscosity at $Re = 4.35 \cdot 10^5$.

Figure 7 shows the pressure distribution $C_p = 2(p - p_0)/\rho u_0^2$ at subcritical, supercritical Reynolds numbers and for the case of an ideal fluid flow. It can be seen that at a supercritical Reynolds number, the pressure distribution approaches the pressure distribution for a vortex-free flow of an ideal fluid, which corresponds to the experimental data [3].

Figure 7. Pressure distribution on the surface of the ball at subcritical and supercritical Reynolds numbers:
1. $Re_{\text{subcrit}} = 1.71 \cdot 10^5$;
2. $Re_{\text{supercrit}} = 4.35 \cdot 10^5$;
3. theoretical.

As an example, calculations of axially symmetric surfaces of various geometries at sub- and supercritical Reynolds numbers are carried out. Figure 8 shows the flow around an ellipse at a compression ratio $k = 0.5$ ($k = b/a$, $a$ and $b$ – the horizontal and vertical axes of the ellipse, respectively, the number $Re$ is calculated by $b$). The impedance coefficient for sub- and supercritical flow regimes is 0.327 and 0.104, respectively. Figure 9 shows the distribution of turbulent viscosity in the vicinity of the ellipse. Figure 10 shows the picture of the flow around the "dumbbell". It can be seen from the figures that at the supercritical Reynolds number, not only the size of the vortex region in the aft part of the body changes, but also the vortex formed in the notch of the upper part of the body degenerates. These areas correspond to the maximum values of turbulent viscosity (figure 11).
Figure 8. Influence of the Reynolds number on the flow pattern around the ellipse.

Figure 9. Distribution of turbulent viscosity related to molecular viscosity $Re = 4.35 \cdot 10^5$.

Figure 10. Influence of the Reynolds number on the flow pattern near the "dumbbell".

Figure 11. Distribution of turbulent viscosity related to molecular viscosity $Re = 4.35 \cdot 10^5$.

6. Conclusion
Calculations have shown that the proposed mathematical model satisfactorily describes the transient processes in the boundary layer on the surface of axially symmetric bodies at the critical value of the Reynolds number.

The mathematical model and the method of numerical implementation can be used to solve the problems of erosional combustion of granular fuels in a turbulent flow, to determine the flow parameters and drag coefficients of axially symmetric bodies, when modeling two-phase turbulent flows in power plants.
Acknowledgements
The article was written with the support of the BGA/20-28-09 grant from the Kalashnikov Izhevsk State Technical University

References
[1] Gorokhov M M, Rusyak I G and Tenenev V A 1996 Izvestiya RAN. Seriya Fluid and Gas Mechanics 4 162
[2] Tenenev V A, Rusyak I G and Gorokhov M M 1997 Math Modeling 9(5) 87-96
[3] Loytsanskiy L G 1987 Fluid and Gas Mechanics (Moscow: Nauka) p 840
[4] Guschin V A 2018 Contemporary Problems of Mechanics and Mathematics dedicated to the 110th anniversary of the birth of S A Khristianovitch (Moscow) 26-7
[5] Sekundov A N 1971 Izvestiya of the Academy of Sciences of the USSR Seriya Fluid and Gas Mechanics 5 114-27
[6] Korepanov A V 2004 Numerical Modeling of Solid Granular Fuel Combustion in Turbulent Flow (Moscow) 106
[7] Kurbatskiy A F 1989 Izvestiya of the Siberian Branch of the USSR Academy of Sciences 6 119-45
[8] Patankar S 1984 Numerical Methods for Solving Heat Transfer and Fluid Dynamics Problems (Moscow: Energoizdat) p 152
[9] Anderson D, Tannehill J and Pletcher R 1990 Computational Fluid Mechanics and Heat Transfer (Moscow: Mir) 2 p 726
[10] Godunov S K, Zabrodin A V and Ivanov M Ya 1967 Computational Mathematics and Mathematical Physics 7(5) 1031-59
[11] Gromadka T and Ley Ch 1990 Complex Boundary Element Method in Engineering Problems (Moscow: Mir) p 303
[12] Sternet L E 1980 Two-Phase Mono- and Polydisperse Gas Flows Containing Particles (Moscow: Mashinostroenie) p 176
[13] Shraiber A A, Lilyutin V N and Yatsenko V P 1980 Fluid Mechanics of Two-Component Flows with Solid Polydisperse Substance (Kiev: Naukova Dumka) p 252
[14] Van-Dyke M 1986 An Album of Fluid and Gas Motion (Moscow: Mir) p 215