Numerical simulation of airflows in a street canyon

R.B. Nuterman and A.V. Starchenko

Tomsk State University

Received January 14, 2003

A steady plane-parallel isothermal turbulent flow of a viscous incompressible liquid above a surface with large-scale roughness is considered. The numerical model includes the Reynolds equations with the Boussinesq closing relations. Turbulent parameters are predicted on the basis of the k–ε model of turbulence and Launder–Spalding's method of wall functions for calculation of near-wall flows. The problem is solved numerically by the finite-volume method. The influence of atmospheric parameters on airflows in a street canyon is investigated. Unfavorable meteorology conditions leading to accumulation of pollutants in urban blocks are revealed.

Introduction

The air quality monitoring system in big cities should supply information both about peak pollution levels in a short period of time and about mean concentrations of atmospheric pollutants. Just the peak levels are largely connected with the pollution associated with traffic. The knowledge of the peak values of pollution and the character of pollution dispersion is very important, since these values often exceed the maximum permissible concentrations.

To describe properly the situation formed by the urban traffic in tunnels, on crossroads, and in urban street canyons, it is necessary to establish a connection between sources and receivers of the urban air pollution. Traffic plays a very important role in the pollution process. Car exhausts strongly contaminate the city environment. The transport emissions are turbulent, and therefore the geometry of a studied region of pollution is of particular importance. A city canyon is formed by buildings, between which local emissions exceed the background. In this situation, the most important problem is to determine the distribution of emissions and their relative contribution to the general pattern. Not only the canyon geometry, but the meteorological conditions also turn out to be of principal importance.

The aim of this paper is to study the effect of the street canyon geometry on the aerodynamic pattern of the turbulent air motion and the distribution of air pollutants emitted by the urban traffic. The mathematical simulation and numerical solution of the problem is conducted.

Physical formulation of the problem

Under consideration is the plane-parallel motion of a viscous incompressible liquid over a surface with a large-scale roughness. The motion is turbulent and isothermal. Roughness elements are rectangular obstacles, whose size is comparable with the size of the region under study.

The velocity profile at the left boundary is described by the following function:

$$u = u_{300}\left(\frac{y - Ly_1}{300 - Ly_1}\right)^{0.3}, \quad y \geq Ly_1,$$

u_{300} is the velocity at the height of 300 m. The motion has a steady character.

Mathematical formulation of the problem

The mathematical formulation of the problem includes the Reynolds equations written with the use of the Boussinesq closing relations[1]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0; \tag{1}$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial \tilde{P}}{\partial x} + 2\frac{\partial}{\partial x}\left[(\nu + \nu_T)\frac{\partial u}{\partial x}\right] + \frac{\partial}{\partial y}\left[(\nu + \nu_T)\frac{\partial u}{\partial y}\right] + \frac{\partial}{\partial y}\left[(\nu + \nu_T)\frac{\partial v}{\partial x}\right], \tag{2}$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial \tilde{P}}{\partial y} + \frac{\partial}{\partial x}\left[(\nu + \nu_T)\frac{\partial v}{\partial x}\right] + 2\frac{\partial}{\partial y}\left[(\nu + \nu_T)\frac{\partial v}{\partial y}\right] + \frac{\partial}{\partial x}\left[(\nu + \nu_T)\frac{\partial u}{\partial y}\right]. \tag{3}$$

Here u and v are the projections of the velocity onto the axes Ox and Oy; ν is the kinematic air viscosity; ν_T is the turbulent viscosity; $\tilde{P} = P + (2\rho k/3)$, where P is pressure; k is the kinetic energy of turbulence; ρ is the air density.

The following boundary conditions for system of equations (1)–(3) are chosen:

– at the left boundary at $x = 0$:

$$u(0, y) = u_{300}\left(\frac{y - Ly_1}{300 - Ly_1}\right)^{0.3};$$

$$v(0, y) = 0;$$

0235-6880/03/05–06 483-04 $02.00

– at the right boundary at $x = Lx$ "soft" boundary conditions of flow stabilization are used:

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0;$$

– at the solid lower boundary, the attachment conditions are used:

$$u = v = 0;$$

– at the top boundary $y = Ly$ it is believed that the velocity components are known:

$$u = u_{300}\left(\frac{y - Ly_1}{300 - Ly_1}\right)^{0.3},$$

$$v = 0.$$

For determination of turbulent parameters of the flow, the k–ε model of turbulence is used[2]:

$$u\frac{\partial k}{\partial x} + v\frac{\partial k}{\partial y} = \frac{\partial}{\partial x}\left[\left(\nu + \frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x}\right] + \frac{\partial}{\partial y}\left[\left(\nu + \frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial y}\right] + P - \varepsilon, \quad (4)$$

$$u\frac{\partial \varepsilon}{\partial x} + v\frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x}\left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right] + \frac{\partial}{\partial y}\left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial y}\right] + (c_1 P - c_2\varepsilon)\frac{\varepsilon}{k}; \quad (5)$$

$$\nu_T = c_\mu k^2 / \varepsilon. \quad (6)$$

Here ε is the dissipation of the turbulence energy k; the turbulence energy generation is described by the equation

$$P = \nu_T\left[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2\right];$$

where the constants are $c_1 = 1.44$, $c_2 = 1.92$, $c_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$.

The boundary conditions for Eqs. (4) and (5) can be written in the following form:

– at the left boundary at $x = 0$:

$$k = k_0(y),\ \ \varepsilon = \varepsilon_0(y);$$

– at the right boundary at $x = Lx$:

$$\frac{\partial k}{\partial x} = \frac{\partial \varepsilon}{\partial x} = 0;$$

– at the top boundary at $y = Ly$:

$$\frac{\partial k}{\partial y} = \frac{\partial \varepsilon}{\partial y} = 0.$$

To specify the values of turbulent parameters near the surface of the roughness elements, the method of wall functions is used.[2] This choice of the method for setting the boundary conditions for k and ε is caused by the fact that turbulent characteristics near the surface (in the buffer layer and viscous sublayer) vary markedly. To describe their behavior using the finite-difference method, a great number of nodes are needed. At the same time, it is known that in the zone of developed turbulence the variation of the tangent velocity as a function of the distance from the surface is well described by the log law, while the turbulence energy is described by the linear law. In this connection, to determine the parameters near walls, we use the Launder–Spalding's method of wall functions,[2] according to which the velocity component tangent to the surface can be represented near the surface as

$$u_\tau = \frac{\tau_w}{\rho c_\mu^{1/4}\kappa k^{1/2}}\ln\left[E c_\mu^{1/4} k^{1/2} n / \nu\right],$$

where $\kappa = 0.42$; $E = 9.0$; τ_w is the surface friction. The kinetic energy of turbulence k near the surface (in the near-wall cell of a difference grid) is determined from Eq. (4) with the use of the following equations for generation and dissipation of the turbulence energy:

$$P = \nu_T\frac{|\tau_w|}{(\kappa n)^2},\ \ \varepsilon = \frac{\left[k c_\mu^{1/2}\right]^{3/2}}{\kappa n},$$

where n is the distance from the streamline surface.

The pollutant concentration field can be determined from solution of the pollutant transport equation having the following form:

$$\frac{\partial(uC)}{\partial x} + \frac{\partial(vC)}{\partial y} = \frac{\partial}{\partial x}\left[\left(\frac{\nu}{\mathrm{Sc}} + \frac{\nu_T}{\mathrm{Sc}_T}\right)\frac{\partial C}{\partial x}\right] + \frac{\partial}{\partial y}\left[\left(\frac{\nu}{\mathrm{Sc}} + \frac{\nu_T}{\mathrm{Sc}_T}\right)\frac{\partial C}{\partial y}\right] + S. \quad (7)$$

Here C is the pollutant concentration; Sc is the Schmidt number; Sc_T is the turbulent Schmidt number; S is the source term. This differential equation was integrated with the zero boundary condition for the pollution concentration at the left boundary and simple gradient relations at the other boundaries.

Solution

The problem formulated is solved numerically on a uniform grid. The differential equations are quantized by the finite-volume method,[3] and the convective terms of the transport equations are approximated using the Van Leer Monotonized Linear Upwind (MLU) scheme.[4] The computational area is formed by a fixed number of nonoverlapping finite volumes in such a way that every node of the computational grid is contained in one volume. After this division of the computational area, the differential equations are integrated over each finite volume. The integrals are calculated with the use of piecewise-linear profiles, which describe the variable change between nodes.[5] As a result of such integration, we get the discrete analog of the differential equations, which includes the values of the variable at several neighboring nodes. For its solution, the method of fictitious areas was used. The essence of this method is that the values of vector and scalar parameters in the obstacle region are zero and diffusion is absent in fictitious finite volumes.

The values of the wind velocity components are determined at the edges of finite volumes, while the characteristics are determined at their centers. For calculation of the flow field, the Patankar–Spalding's SIMPLE procedure[3] was used. This procedure involves the following operations:

1. Setting initial approximations for all dependent variables.
2. Solving the equations of momentum variation to determine tentative values of the longitudinal and cross velocity components.
3. Solving the Poisson equation for pressure correction.
4. Determining the new pressure field.
5. Correcting the velocities.
6. Solving the discrete analogs for the turbulent characteristics and the concentration.
7. Taking the obtained values of the dependent variables as initial ones and repeating all the operation starting from the second one.

Results and discussion

To determine the character of the air motion and the distribution of the pollution concentration in the street canyon, as well as to find the dependence of the concentration on the canyon geometry, a series of computations was carried out. The computations were performed on a 81×81 grid. Two pollution sources of constant intensity were located near the surface ($y = 0$) 1 m far from the vertical side walls of the canyon. Figures 1–5 depict the results of the computation, as well as the vector fields and concentrations reconstructed from them. The air speed was 3 m/s, and the pollution emission intensity was constant.

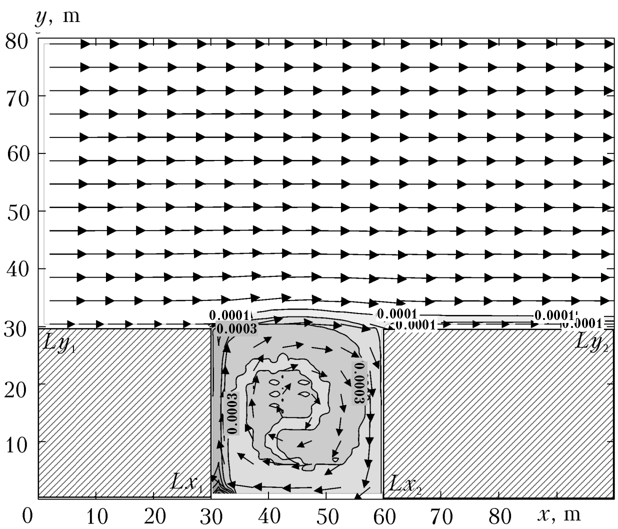

Fig. 1. Vector field of wind velocity and pollution distribution in the street canyon. $Ly_1 = Ly_2 = 30$ m; $Lx_2 - Lx_1 = 30$ m.

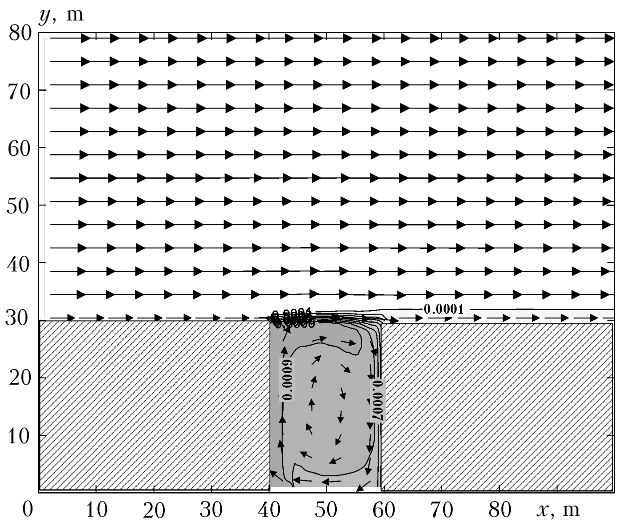

Fig. 2. The same as in Fig. 1 at $Ly_1 = Ly_2 = 30$ m; $Lx_2 - Lx_1 = 20$ m.

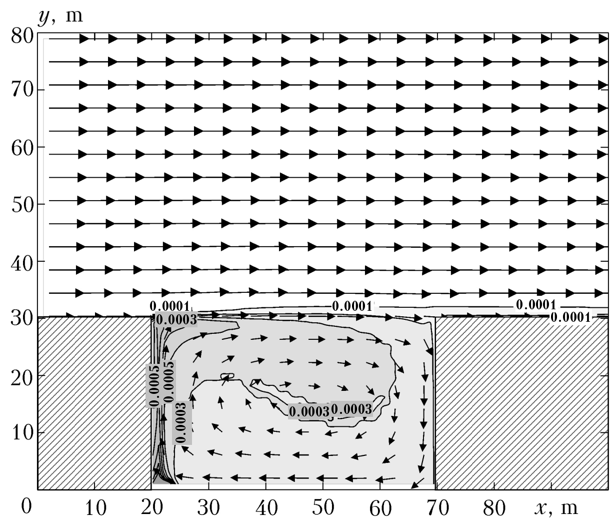

Fig. 3. The same as in Fig. 1 at $Ly_1 = Ly_2 = 30$ m; $Lx_2 - Lx_1 = 50$ m.

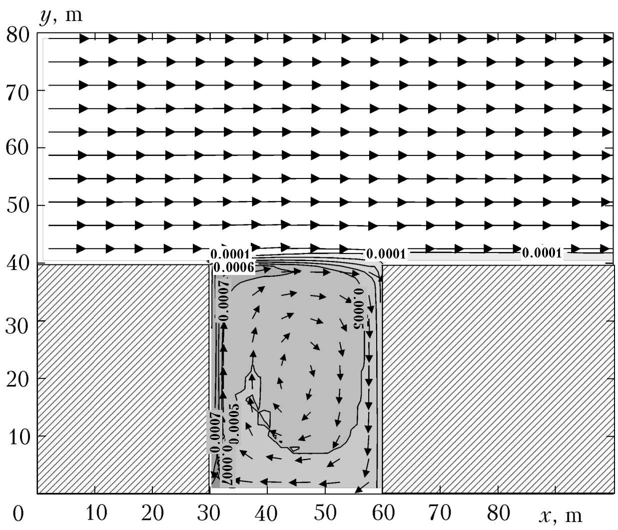

Fig. 4. The same as in Fig. 1 at $Ly_1 = Ly_2 = 40$ m; $Lx_2 - Lx_1 = 30$ m.

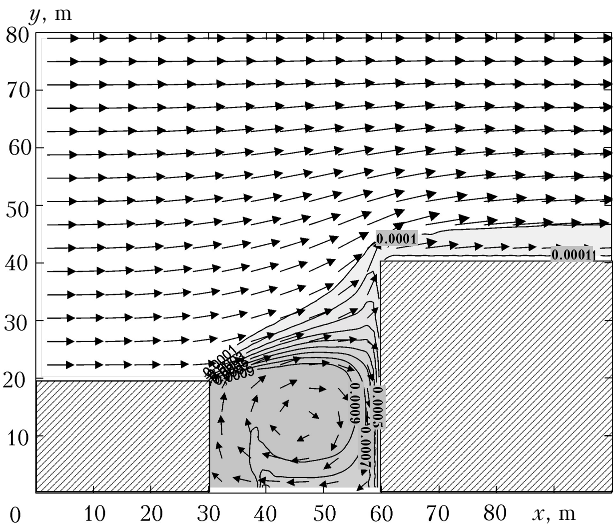

Fig. 5. The same as in Fig. 1 at $Ly_1 = 20$ m; $Ly_2 = 40$ m; $Lx_2 - Lx_1 = 30$ m.

It can be seen from Figs. 1–5 that the presence of a canyon formed by neighboring buildings affects the atmospheric air motion over the city. In all considered cases of the canyon geometry, circulation motion with pollution removal from the canyon was observed. The change in the canyon volume affects the magnitude of the mean concentration, but in all the cases local peaks of the pollution concentration were observed near the lee side of the canyon.

As follows from Figs. 2 and 4, the increase of the canyon depth or its narrowing leads to some increase of concentration of pollutants. It is important to note that the concentration also increases in the case, if the building on the lee side is lower than the next one (see Fig. 5).

Conclusion

In this paper, we have described the mathematical model and the computational method for investigation of aerodynamics in a street canyon. Parametric computations have been carried out to reveal the effect of the canyon depth, width, and shape on air motion in it and the distribution of pollutants emitted by continuous point sources located at the canyon bottom.

References

1. L.G. Loitsyanskii, *Liquid and Gas Mechanics* (Nauka, Moscow, 1987), 840 pp.
2. B.E. Launder and D.B. Spalding, Computational Methods in Applied Mechanics and Engineering **3**, No. 2, 269–289 (1974).
3. S.V. Patankar, *Numerical Heat Transfer and Fluid Flow* (Hemisphere Publishing Corporation, New York, 1980).
4. B. Noll, AIAA Journal **30**, No. 1, 64–69 (1992).
5. A.O. Esaulov and A.V. Starchenko, in: *Computational Hydrodynamics* (Tomsk State University Publishing House, Tomsk, 1999), pp. 27–32.