Large Eddy Simulation of Turbulent Flow and of Pollutant Transport in a Street Canyon

Alexander V. Starchenko*[a,b], Evgeniy A. Danilkin[a]
[a]Tomsk State University, 36 Lenin Avenue, Tomsk, Russia, 634050;
[b]Institute of Monitoring of Climatic and Ecological Systems SB RAS, 10/3, Akademichesky ave. Tomsk, Russia, 634055

ABSTRACT

The work presents a non-steady three-dimensional eddy resolving model intended for the simulation of non-isothermal turbulent separation flows in street canyons. For a subgrid-scale turbulence parameterization, the Smagorinsky gradient model is used. The calculation results demonstrate the effects of pollutant source location, street canyon size, basic stream rate and wall temperature difference on air pollution in the canyon.

Keywords: Large Eddy Simulation, Turbulence, Street Canyon

1. INTRODUCTION

Due to the continuously growing amount of motor transport the pollution of the atmospheric boundary layer with exhaust fumes also grows. In order to provide a high level of ecological comfort for the population while efficiently using urban land, the instrumental control of atmospheric air composition should be accompanied by the use of environmental monitoring systems capable of predicting the concentration of gas components of pollutants under various meteorological conditions and configurations of urban built-up areas. For this purpose there are quite popular mathematical modelling methods that make it possible to calculate the detailed structure of turbulent flow of air masses and the pollutants distribution pattern as well as to predict the areas with pollutants exceeding the maximum permissible concentration at specific urban built-up areas [1], [2], [3], [4], [5].

At the current development stage of turbulence theory the environmental turbulent flows are modelled mainly using Reynolds Averaged Navier-Stokes and transport equations – (RANS) that require a solution of closure problem by involving semi-empiric models of different levels of complexity. However, despite quite successful development of these models they can hardly describe non-stationary turbulent flows in the vicinity of blunt bodies. First of all, it is attributable to the slipstreams specifics, i.e., to the availability of organized coherent structures determined by the flow parameters and research area geometry [6], [7]. Large Eddy Simulation – (LES) [7], [8] appears to be preferable for modelling turbulent slipstreams as it permits to describe the non-stationary structure of a turbulent flow and to directly predict the behavior of large eddies and the cascade process of energy transfer to smaller eddies with dimensions as small as the grid cell. In this case smaller scaled eddies that can be regarded as isotropic are modelled using a particular subgrid model.

This work deals with the definition and assessment of a three-dimensional eddy-resolving microscale mathematic model that takes into account the impact of the substrate buoyancy and roughness effects in the investigation of turbulent slipstreams and of processes of pollutant transport in street canyons. It is of interest to obtain new information about the impact of thermal conditions on the flow structure and to investigate local specifics of changes in the temperature as well as formation of highly polluted atmospheric zones in urban quarters.

2. FORMULATION PROBLEM

Physical Formulation

A three-dimensional non-steady turbulent air flow in a street canyon is considered. Pollution sources of constant intensity are located at the street canyon floor. The simulation domain is shown in Fig. 1.

21st International Symposium on Atmospheric and Ocean Optics: Atmospheric Physics,
edited by G. G. Matvienko, O. A. Romanovskii, Proc. of SPIE Vol. 9680, 968062

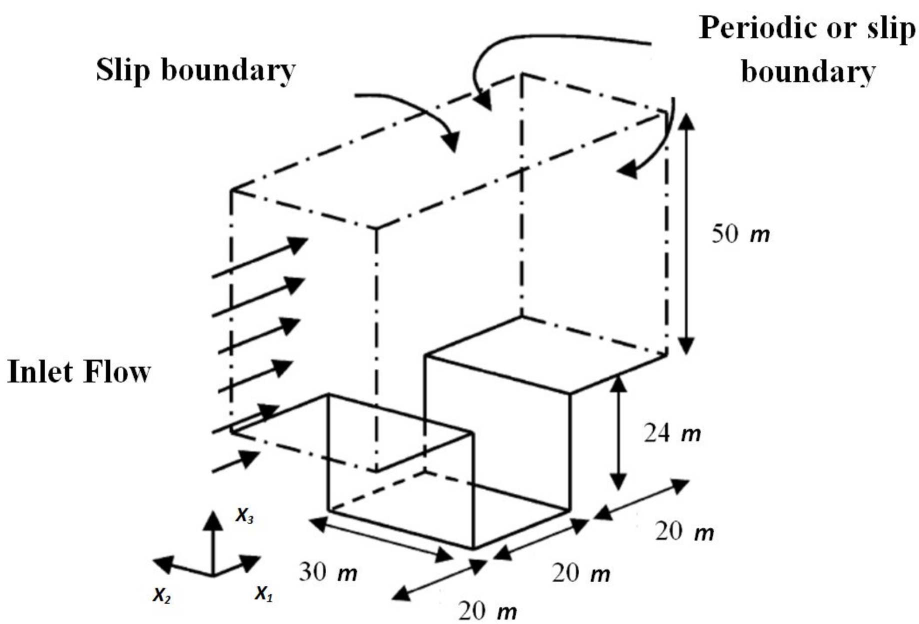


Figure 1. Simulation domain (24 meter-deep street canyon).

Mathematical Formulation

The turbulent flow and transport modelling is made with the large eddy method, which basic idea is a formal mathematical separation of large and small eddies by means of high-frequency filters. The large eddies are resolved explicitly while small-scale turbulence is parameterized, with the large-scale eddies characteristics. This possibility is provided by the Kolmogorov universal equilibrium theory [9] according to which if it is impossible to resolve all turbulent motion scales in a numerical simulation, then the small-scale eddies should be modelled as approximately isotropic structures in contrast to the calculated apparently anisotropic large-scale eddies. This work uses implied filtration of the Navier-Stokes equations where the difference grid used serves as a filter [10].

The mathematic model of three-dimensional non-stationary turbulent flow of an incompressible flowcomprises the filtered continuity equations and the Navier-Stokes equations in the Boussinesq approximation [8]:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0; \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial}{\partial x_j}\left(\frac{\partial \overline{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} + g_i \frac{\overline{T} - T_0}{T_0}, i = 1,2,3. \tag{2}$$

There $\overline{u}_i$ are filtered projections of instantaneous velocity vector on the axis of coordinates, $\overline{p}, \overline{T}$ are filtered values of pressure and temperature, T_0 is the relative value of temperature; ν is the kinematic viscosity, ρ is density, τ_{ij} is subgrid turbulent stresses tensor, g_i are projections of the gravity vector (0;0;9,81). For the recurrent indexes j totaling is made.

The system of equations obtained is incomplete and can be closed using the Smagorinsky subgrid model [11]:

$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} \approx \tau_{ij}^{smag} = -\nu_T \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right),\ i, j = 1,2,3; \tag{3}$$

where ν_T is turbulent viscosity. Turbulent viscosity ν_T is defined by the mean dissipation rate of turbulent energy per unit volume and by the scale of subgrid eddies [8]:

$$\nu_T = C_s^2 \Delta_g^2 \left|\overline{S}\right|; \tag{4}$$

where C_s is the Smagorinsky constant, $\Delta_g = h$ is the model grid step, $\left|\overline{S}\right| = \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$ is the worm of the strain rate tensor.

The mathematical model also comprises filtered transfer equations that describe the transport of pollutant concentration and heat:

$$\frac{\partial \overline{C}}{\partial t} + \frac{\partial \overline{u}_j \overline{C}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\frac{\nu}{Sc} + \frac{\nu_T}{Sc_\tau}\right)\frac{\partial \overline{C}}{\partial x_j}\right) + S_{pol}; \tag{5}$$

$$\frac{\partial \overline{T}}{\partial t} + \frac{\partial \overline{u}_j \overline{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\frac{\nu}{\Pr} + \frac{\nu_T}{\Pr_\tau}\right)\frac{\partial \overline{T}}{\partial x_j}\right) + S_{temp}; \tag{6}$$

where $\overline{C}$ is the pollutant concentration, Sc is Schmidt number, Sc_τ is turbulent Schmidt number, $\overline{T}$ is the temperature field, $\Pr$ is Prandtl number, $\Pr_\tau$ is turbulent Prandtl number, S is the function describing sources distribution.

Boundary and Initial Conditions

Conditions of no-slip and impermeability on the solid interface were used as boundary conditions for the velocity. At the start time a velocity profile was set at the inlet and zero normal derivative condition was set at the outlet, then in the course of calculation a flow periodicity condition was used in the flow direction. For the pollutant concentration, on the walls and at the outlet zero normal derivative condition was set and zero values were set at the inlet.

Since the present spatial resolution does not make it possible to describe the processes taking place in the viscous boundary layer, the total action of the wall should be taken into account using a near-wall model. It is well known that in a fully-developed turbulence zone, the change in the longitudinal velocity component with the distance from the surface can be rather accurately approximated by a logarithmic dependence. For this reason, in order to describe the mean values' behavior correctly, this work used a very simple near-wall model well within the limits of the first calculation layer; i.e., a model that was successful in other works and is described in [10].

Ignoring the velocity profile deviation from the logarithmic law, the velocity in the first calculation node above a rough surface can be estimated using the following formula:

$$\overline{u} = \frac{\overline{u}_\tau}{k}\ln\left(\frac{z}{z_0}\right); \tag{7}$$

where $\overline{u}_\tau = \sqrt{\tau_w / \rho}$ is the dynamic velocity, τ_w is friction stresses on the wall, z_0 is roughness parameter, $k \approx 0,4$ is the Karman constant, z is the distance to the surface. The hypothesis about a logarithmic distribution of the velocity profile was used when assessing the influence of friction stresses at the wall.

3. DIFFERENCE APPROXIMATION AND NUMERICAL SOLUTION

The above obtained differential equation system in partial derivatives is solved numerically by a finite volume method using a staggered difference mesh. The convection terms are approximated using Van Leer's upwind scheme MLU. The diffusion terms are approximated by a central difference scheme of second order of approximation. The motion equations are solved using the Adams-Bashford's explicit scheme. The resultant second-order difference scheme of time and space approximation is conditionally stable.

In the hydrodynamic part of the model, the fields of velocity and pressure are correlated using a predictor-corrector scheme, in which the Adams-Bashford's explicit scheme for the motion equation performs a function of predictor while the velocity field is corrected by the solution of a Poisson equation for the pressure using a conjugate gradient (CG) method preconditioned by a Seidel method with red-and-black arrangement.

In this work, the flows in areas of complicated geometry were calculated using a fictitious domain method whose principle is that the values of the vector and scalar variables in the obstacle area are equal to zero and there are no diffuse flows at the boundaries of fictitious final volumes.

The mathematic model and the parallel MPI-program have been checked on a number of test cases that involved a flow around a square-section cylinder, and a flow over a rough plate [12].

4. SIMULATION RESULTS

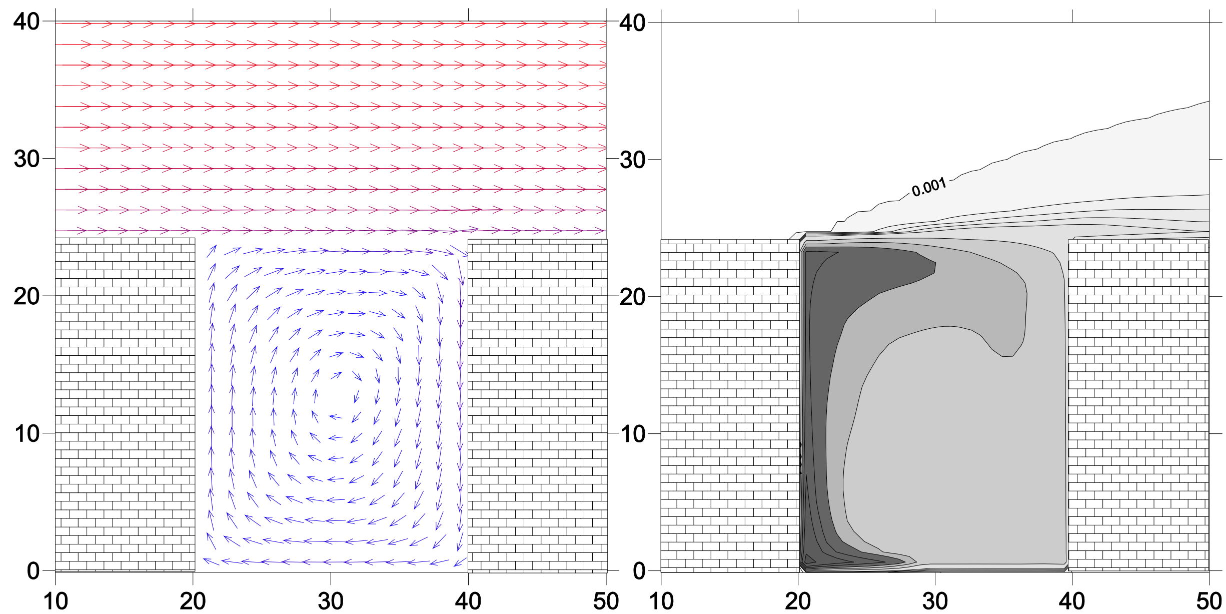
Based on our mathematical model of turbulent flow of incompressible medium a number of calculations have been performed with the three-dimensional model of a street canyon. The work investigates the nature of pollution propagation depending on the correlation of width and height of a street canyon, depending on basic flow velocity and on temperature inhomogeneity. At the current stage of investigation the temperature inhomogeneity was modelled as follows: the temperature of basic flow was set at 20^{o}C, the street canyon buildings roofs were adiabatic, the temperature of one of the walls or floor was 25^{o}C.

The problem was solved under the following scenario. On the lateral surfaces, in the direction transversal to the main flow, slip conditions were used. In this case it is necessary to place lateral boundaries of computational domain in such a way that the domain length in the transversal direction is larger than the size of the largest eddy in the canyon. For the case under consideration the largest eddy is limited by the canyon width $W = 20$ m while the transversal length of the canyon can be taken equal to 30м. Periodical boundary conditions were set in the longitudinal direction in order to simulate an endless series of canyons. The calculations were performed on a 182x54x180 mesh. A pollutant source of constant rate was located near the surface at $h_z = 0,125$ m height above the base center of the computation domain. The calculations (and values smoothing) were performed for 48 periods after the initialization time equal to 40 periods. The duration of one period was calculated by the formula $T = L/U$, where L is a characteristic dimension, U is a characteristic velocity (5 m/s).

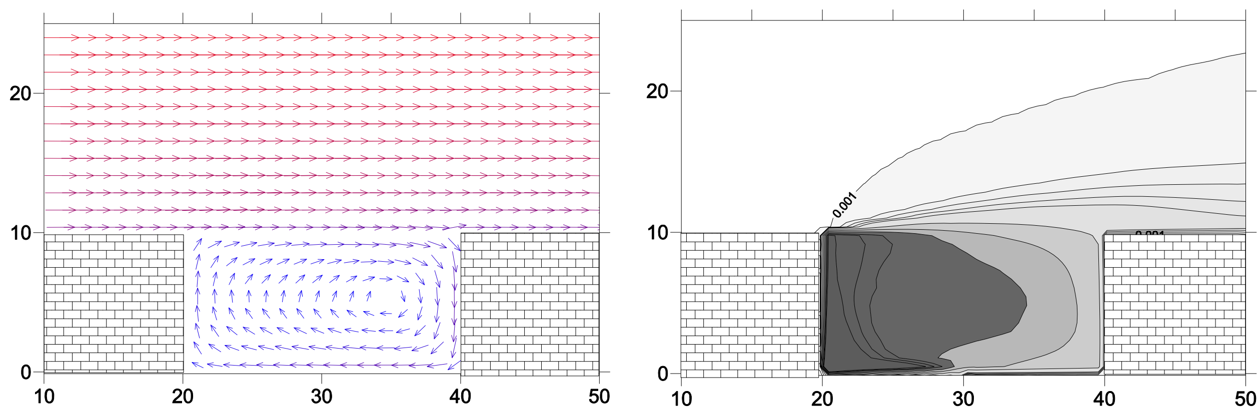
Modelling of air motion for the case under investigation demonstrated that the basic flow involves in a rotatory motion the air masses inside the street canyon (Figure 2) and determines the direction and intensity of the resultant eddy motion. The pollutant coming from the sources on the canyon floor is carried by the rotary motion developed to the leeward side of the canyon and then it partially wears off entering the basic flow and is partially returned, by a rotary motion of air to a domain restricted by the vertical walls of closely spaced buildings.

The numerical experiments have demonstrated that an increase in the basic flow speed providesa more rapid blowout of pollutant from the street canyon. They also demonstrate how street canyon geometry influences the flow shape and pollution level. Thus, when the street canyon height H gets smaller or its width W increases and reaches the $H / W = 0,5$ ratio, the primary eddy center is shifted to the windward building and the eddy stretches along the whole of the passageway length (Fig. 2 b). It results in a reduced velocity of rotating motion of air masses at the leeward side of the canyon. Consequently, the pollutant is blown out of the canyon less rapidly and the local pollutant concentration levels increase. A further increase in the distance between the buildings, $H / W = 0,125$ results in the formation of two recirculation zones: large eddy at the leeward side and a small one at the windward side. In this case the pollutant coming from the source located in the center of the street canyon is carried in the direction of the leeward building where it is rotates in a turbulent vortex.

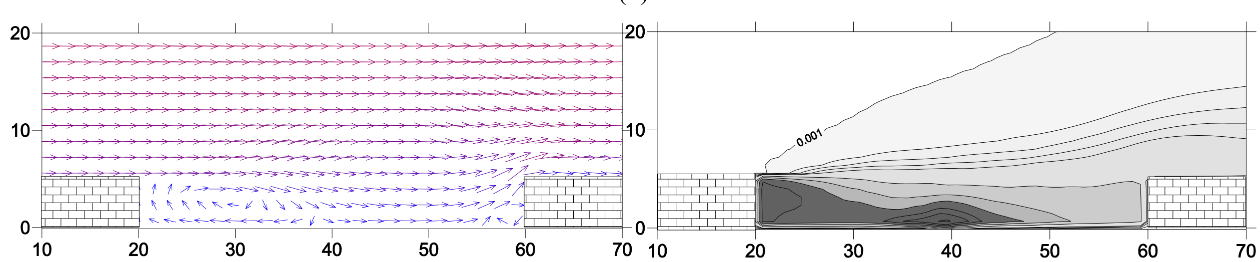
For the initial geometry of a street canyon an investigation was performed to determine the temperature effects on the turbulent flow inside the street canyon. Consequently, three different thermal states of the system were modelled. In the first case the windward side temperature was 320K. In the second case the temperature of the street canyon floor was 320K. In the third case the leeward side temperature was 320K. The temperature of all other surfaces and of the approaching flow was 300K.



(a)



(b)



(c)

Figure 2. Time-averaged vector field of velocity and concentration profile in a street canyon.

The computation results demonstrate that irrespective of the speed range or of temperature distribution on the canyon walls the peak concentration of pollutants is observed on the canyon leeward side and near the pollutant sources (Fig. 2), and at the same time peak concentrations increase when the pollutant source moves to any of the street canyon structural elements.

In the case when the leeward side is heated, the upward flow near the heated leeward wall of the building intensifies the basic vortex and increases the mean velocity of circular motion, so the basic eddy boundaries are consequently widened and grow higher than the street canyon roofs. The heated street canyon floor also intensifies the basic eddy but in this case the vector field of velocity is similar to the case when the walls are heated uniformly and the basic eddy boundaries are at the roof level (Fig.2). In the case when the windward side is heated, an upward flow of air is formed at the windward side. This upward flow moves towards the side opposite to the basic eddy flow. The basic eddy therefore gets compressed and a new smaller eddy is formed between the basic eddy and the leeward side of the building. The results obtained are in good agreement with the mathematical modelling results obtained in pare [1].

5. CONCLUSIONS

In order to investigate the flow structure and pollutant transport processes in street canyons, an eddy resolving model of turbulence that makes allowance for temperature nonuniformity as well as surface roughness has been developed.

Based on the above model the impact of pollutant source location, street canyon geometry, basic flow velocity, and temperature nonuniformity (heated walls) on the pollution level has been investigated.

REFERENCES

[1] Park, S.B., Baik, J.J., Raasch, S., Letzel, M.O., "A large-eddy simulation study of thermal effects on turbulent flow and dispersion in and above a street canyon," J. Appl. Meteor. Climatol. 51(5), 829-841 (2012).
[2] Nuterman R., Starchenko A., Baklanov A., "Numerical model of urban aerodynamics and pollution dispersion," International Journal of Environment and Pollution 44(1-4), 385-393 (2011).
[3] Liu, C.H., Barth, M.C., "Large-eddy Simulation of Flow and Scalar Transport in a Modeled Street Canyon," Journal of Applied Meteorology 41, 660-673 (2002).
[4] Walton, A., Cheng, A.Y.S., "Large-eddy simulation of pollution dispersion in an urban street canyon. Part 2: idealised canyon imulation," Atmospheric Environment 36, 3615-3627 (2002).
[5] Hoydysh, W.G., Dabberdt, W.F., "Kenematics and dispersion characteristics of flows in asymmetric street canyons," Atmospheric Environment 22, 2677-2689 (1988).
[6] Garbaruk, A.V., Strelets, M.H., Shur, M.L. [Turbulence Modelling in Computation of Complex Flows], Polytechnical University Press, St. Petersburg, 88 (2012).
[7] Khlopkov, Yu.I., [Lectures on Theoretical Methods of Investigating Turbulence], Moscow Institute of Physics and Technology, Moscow, 178 (2005).
[8] Sagaut, P., [Large eddy simulation for Incompressible Flow], Springer-Verlag, 556 (2006).
[9] Kolmogorov, A.N., "Local structure of turbulence in the incompressible viscous liquids at very high Reynolds numbers," DAN SSSR 30, 299–303 (1941).
[10] Glazunov, A.V., "Large-eddy Simulation of Turbulence with the Use of a Mixed Dynamic Localized Closure" News of the Russian Academy of Sciences Atmospheric and Oceanic Physics 45(1), 7–28 (2009).
[11] Smagorinsky, J., "General circulation experiments with the primitive equations. I: The basic experiment," Monthly Weather Review 91(3), 99–165 (1963).
[12] Danilkin, E.A., Starchenko, A.V., "High Performance Computation for Large Eddy Simulation," LNCS series 6083, 163-172 (2010).