Assessment of Influence of Residential Area Planning Structure on Air Quality Using Numerical Modeling

Svetlana Valger 1, Anastasia Maslova2

1 Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, 4/1 Institutskaya str., Novosibirsk 630090, Russia
2 Novosibirsk State University of Architecture and Civil Engineering, 113 Leningradskaya str., Novosibirsk 630008, Russia

Valger@itam.nsc.ru

Abstract. Numerical study of the aeration of a new urban microdistrict located along a highway in Novosibirsk, Russia, was performed. The main goal of the paper was to assess whether the architectural and planning structure of the new microdistrict is optimal in terms of aeration and air quality. A scenario of particulate matter (PM10) transfer in the vicinity of the microdistrict is reproduced. Numerical modeling is performed for the prevailing wind direction. 3D numerical simulation was carried out using the Revit software to build the CAD model of the residential area and the ANSYS software was used to perform a computational experiment. Based on the calculations, numerical data were obtained on the wind speed and on PM10 concentration fields in the residential area. Favorable and unfavorable zones of the microdistrict in terms of high concentrations of PM10 at the pedestrian zones were described.

1. Introduction

Modern cities are characterized by a complex architectural and city planning structure that changes dynamically over time. Under the influence of urbanization processes in large cities, the city planning structure is gradually changing. For instance, construction of new micro-districts of high-rise buildings in already existing low-rise and densely-built areas leads to change in urban landscape and in some cases it can lead to dramatic consequences in terms of area ventilation and accumulation of harmful gases and dust particles in residential areas. The location of residential areas in the vicinity of highways can also lead to deterioration in air quality due to the presence of local sources of aerosols emitted by car traffic on the roads. We have an urgent task to develop and validate methods that would allow to assess the impact of city planning structure of buildings on air quality at the stage of residential areas design.

Today, computational fluid dynamics methods (CFD) allow to perform virtual experiments to assess wind loads on buildings and structures, to analyse the area pedestrian comfort as well as to evaluate the aeration of building complexes. Numerical models of fluid mechanics make it possible to study complex processes of air heat- and mass transfer, to detect zones of high aerosol concentrations, to evaluate the effectiveness of various protective measures, such as vegetation zones, protective screens, etc. Using CAD models of buildings in conjunction with Computer-aided engineering software (CAE), we can research various aeration scenarios and optimize architectural and planning solutions for buildings.

In this paper, we performed a numerical study of the aeration of a new urban microdistrict located on the banks of a river and along a busy highway in Novosibirsk, Russia (figure 1,a,b). This case presents an interest for researchers because on the one hand, the considered territory is characterized by strong...
winds, and on the other hand, the territory is located in a zone of heavy traffic. The main goal of the paper was to assess whether the architectural and planning structure of the new microdistrict is optimal in terms of aeration and air quality. Numerical modeling is performed for the prevailing wind direction. Modeling was carried out in 3D approximation using the Revit software to build the CAD model of the residential area and the ANSYS software was employed to perform a computational experiment.

Based on the results of the study, numerical data were obtained on the wind speed fields in the residential area and on the aerosol concentrations in the vicinity of the buildings. On the basis of the calculated data, a scenario corresponding to a medium congestion of a section of road were analyzed and the territory plan was assessed regarding air quality in residential areas. Favorable and unfavorable zones of the microdistrict in terms of high concentrations of PM10 at the pedestrian zones were described.

![Figure 1](image1)

Figure 1. The considered microdistrict, Novosibirsk, Russia.

2. Problem statement
A fragment of the microdistrict located on the bank of the Ob River is considered in the paper (Fig. 2, a). The microdistrict is located along a major highway (Fig. 2, b, Road #1). There is an additional road inside the microdistrict (Fig. 2, b, Road #2) with less intensive traffic. The aim of this study is to model a scenario corresponding to a medium congestion of a section of road #2 and to assess the favorable and unfavorable zones of the microdistrict in terms of high concentrations of PM10 for this case. The south wind direction, which is often repeated for the considered area, and average velocity \( U_h = 3.9 \, \text{m/s} \) at a height \( h = 10 \, \text{m} \) are chosen in the case. The Reynolds number \( Re_L \) calculated using the characteristic wind speed \( U_L = 4.98 \, \text{m/s} \) at the height \( L = 93 \, \text{m} \) of the tallest building of the building complex is \( \approx 3.17 \times 10^7 \).

![Figure 2](image2)

Figure 2. Plan of the microdistrict (a) and fragment of the computational domain in plan (b).
3. Models & Methods

Numerical simulation was performed using the computational domain with dimensions \( L \times W \times H = 2.5 \, \text{km} \times 2.5 \, \text{km} \times 0.5 \, \text{km} \). The buildings have a real scale 1:1 and any details of facades and roofs are not considered in simulation (figure 3, a). The computational domain includes also the roads located in the microdistrict as shown in figure 3, a. At the first stage, it was decided not to take the terrain into account in the calculations. The chosen problem statement does not take into account the small vegetation zones located in the microdistrict. In figure 3, b, c, fragments of the computational grid of finite volumes are shown. The characteristic size of the grid element was \( \Delta \approx 2.5 \, \text{m} \) on the building and \( \Delta \approx 5 \, \text{m} \) in the vicinity of the buildings. In accordance with [1], at least 5 grid element layers were provided below the pedestrian height \( h_p \approx 1.75 \, \text{m} \). The total number of grid elements was about 10 million.

![Figure 3](image_url)

**Figure 3.** 3D models of buildings (a) and a fragment of the computational grid (b).

The boundary conditions at the inlet of the computational domain are specified in accordance with [1]. The profiles of velocity \( U \), turbulent kinetic energy \( k \) (TKE) and dissipation of TKE \( \varepsilon \) are calculated as:

\[
U(z) = U_s \left( \frac{z}{z_s} \right)^\alpha;
\]

\[
I(z) = 0.1 \left( \frac{z}{z_G} \right)^{(-0.05)}; k(z) = (I(z)U(z))^2;
\]

\[
\varepsilon(z) = C_{\mu} k(z) \frac{U_s}{z_s} \alpha \left( \frac{z}{z_s} \right)^{(\alpha-1)},
\]

where \( C_{\mu} = 0.09 \) is the model constant, \( U_s = 3.9 \, \text{m/s} \) is the velocity at reference height \( z_r = 10 \, \text{m} \), \( \alpha = 0.11 \) is the power-law exponent for the open sea [2] and \( z_s \) is the boundary layer height.

Soot particles of density \( \rho_p = 2 \, \text{g/cm}^3 \) and diameter \( d_p = 10 \, \mu\text{m} \) enter to the computational domain at the boundary Road #2 (figure 2, b). The particle mass flow rate trough the fragment of the road is \( \approx 7.33 \times 10^4 \, \text{kg/s} \). It describes the scenario of car flow rate \( Q = 1200 \) vehicles per hour under conditions of emission of PM10 \( \approx 0.025 \, \text{g/km} \) from the vehicle.

The symmetry condition is specified at the side and top boundaries of the computational domain and the pressure outlet condition \( \Delta P = P_s - P_0 = 0 \, \text{atm} \) is specified at the outlet boundary. The thermal effects are neglected in the flow that corresponds to the neutral thermal regime for the ambient air.

The computational domain with LOD100 was filtered and exported from the CAD Revit to ANSYS 2020R1. The ANSYS Fluent 2020R1 solver was used to perform the calculations. A flow includes two
phases corresponding to soot particles and air. The 3D Reynolds-averaged Navier-Stokes equations are used to describe the air flow. The Discrete Phase Model (DPM) is used to describe the motion of soot particles [3]. This model allows to describe the motion of particles based on the equation of the force balance on the particle integrated in the Lagrangian frame. The model follows the assumptions: the particles are spherical and they are considered as points located in the center of the spheres; the density of particles is significantly higher than the density of the continuous phase; particle interaction is neglected; particle motion does not affect the turbulence parameters of the continuous phase. The dispersion of particles due to fluid turbulence is predicted using the stochastic tracking model (DRW) [4]. To simulate turbulent effects in the air, the k-ε RNG turbulence model [5] is used, which has proved satisfactorily in the calculations of aerodynamics of bluff bodies [6,7].

Pressure-velocity coupling method was chosen to solve equations of the continuum phase and the Second Oder Upwind scheme was used for the convective terms. The calculations were performed in steady statement with an accuracy of 10⁻⁴ for all equations.

4. Results and discussions
The aim of this study is to model a scenario corresponding to a medium congestion of a section of road #2 and to assess the favorable and unfavorable zones of the microdistrict in terms of high concentrations of PM10 for this case.

At the first step, the aeration conditions of the microdistrict were investigated using the calculated velocity distribution at the pedestrian level height. Figure 4 shows the velocity field at the height $z_L=1.75$ m over the ground surface.

Buildings of the microdistrict can be classified for three main zones relatively the considered wind direction. Zone #1 includes buildings located on the first line from the river. Almost all buildings have enclosed courtyards with playgrounds and recreation areas in the Zone #1. Zone #1 also includes three high-rise buildings (1-3). Zone #2 is located on the second line from the river behind highway 2 relatively the south wind direction. There are no high-rise buildings in the Zone #2 and the buildings are also characterized by enclosed courtyards. Zone #3 is located at a distance from the river and is characterized by the presence of several high-rise buildings without courtyard areas.

In figure 4, arrows schematically indicate areas with high wind speed. Typically, these areas are located between buildings and form narrow gaps in which the flow is accelerated. In most of these zones, low-traffic roads with pedestrian zones were build. Stagnation zones (a) with low air velocity are formed behind buildings in Zone #1, and spread to Road #2, located between Zones #1 and #2. In Zone #2, an extensive recirculation zone (b) is also formed. In the zone (b), recreation areas and children's educational institutions are planning in the future. The Zone #3 is located in the shadow area behind Zones #1 and #2 and high wind velocity is observed only in area (c).

At the second step, the distribution of PM10 concentrations in the computational domain is investigated. Figure 5 shows the isosurface of PM10 concentration $C = 1 \mu g/m^3$. Figure 6 shows the contour of PM10 concentration in horizontal plane at the height $z = 1.75$ m. As we can see from the figures, high PM concentrations are observed in the recirculation zones behind the buildings in Zone #1 (a). It can be seen in the figure 5 that in the recirculation zones behind the buildings located on the first line from the river, the particle cloud rises to a height comparable to the height of the buildings. Particles in these zones are entrained by the flow into the recirculation zone and do not move away with the external flow. In zone (b), by contrast, a large recirculation zone displaces the particle cloud and it leads to the displacement of the high concentration zone close to the road. In zones with good ventilation (for example, zone c, Fig. 4), particles are concentrated along the ground surface and flow downstream. Pigh concentrations of particles are also observed in zone (d), figure 6. This zone contains a stagnant flow area. There is a kindergarten in zone (d) in the microdistrict what can have a negative effect under the conditions of the considered scenario.
Figure 4. Velocity field $U_{z0}$ at the section $z_0=1.75$ m.

Figure 5. Isosurface of PM10 concentration $C=1$ μg/m$^3$. 
5. Conclusions
A numerical simulation of the scenario of the PM10 transfer in the vicinity of a new urban microdistrict located in Novosibirsk, Russia was performed. The numerical modeling was carried out for the prevailing wind direction and took into account the medium congestion of the road located in the microdistrict. Modeling made it possible to evaluate the air flow structure around the building complex, to analyze zones with good ventilation and stagnant zones. The analysis of the particle concentration field at the height of 1.75 m above the ground showed that there are unfavorable pedestrian areas in terms of air quality in the microdistrict. The several recreation areas are located in unfavorable zones. Redistribution of particle concentrations can be achieved by creating green and vegetation zones, as well as the location of small architectural forms. Further work will focus on exploring other particle transport scenarios and developing recommendations for improving aeration of the territory.

Acknowledgment
The research was supported by the Russian Science Foundation (project No. 20-79-00151).

Reference
[1] Y. Tominaga et al., “AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings,” *J. Wind Eng. Ind. Aerodyn.*, vol. 96 (10–11), pp. 1749–1761, 2008.
[2] S. Hsu, E. Meindl, D.B. Gilhousen, “Determining the Power-Law Wind-Profile Exponent under Near-Neutral Stability Conditions at Sea,” *J. Appl. Meteorol. Climatol.*, vol. 33 (6), pp. 757–765, 1994.
[3] ANSYS Fluent Theory Guide, Release 2020R1, ANSYS Inc. (2020)
[4] A. D. Gosman and E. Ioannides “Aspects of computer simulation of liquid-fueled combustors,” *J. Energy*, vol. 7 (6), pp. 482–490, 1983.
[5] V. Yakhot, S. A. Orszag, “Renormalization group analysis of turbulence. I. Basic theory,” *Journal of Scientific Computing*, vol. 1(1), pp. 3–51, 1986.

[6] J. Liu, J. Niu, “CFD simulation of the wind environment around an isolated high-rise building: An evaluation of SRANS, LES and DES models,” *Building and Environment*, vol. 96, pp. 91–106, 2016.

[7] Y. Tominaga, T. Stathopoulos, “Numerical simulation of dispersion around an isolated cubic building: comparison of various types of k-ε models,” *Atmos. Environ.*, vol. 43, pp. 3200-3210, 2009.