Belső udvarok légszennyezettségének modellezése különböző peremfeltételek és szélsebességek esetén városi környezetben

A simulation study to quantify concentration of air pollutant in urban environment – courtyards - using different boundary conditions and wind speed

Katalin Antalné Bökfi[1], Tamás Varga[2], Endre Domokos[1]

[1]Institute of Environmental Engineering, Faculty of Engineering, University of Pannonia, 10 Egyetem St., Veszprem, Hungary H-8200

[2]Department of Process Engineering, Faculty of Engineering, University of Pannonia, 10 Egyetem St., Veszprem, Hungary H-8200

Összefoglaló

Manapság a légszennyezés az egyik legmeghatározóbb tényezője a városi környezetnek. A városi levegő minőségének alakulását számos tényező befolyásolja. Jelen tanulmány a közlekedési eredetű légszennyezők vizsgálatára fókuszál. Számos tanulmány foglalkozik utcakanyon modellek alkotásával, azonban a belső udvarok, melyre ebben a tanulmányban koncentrálunk, kiemelten fontos és kevésbé tanulmányozott részei a városi környezetnek. A belső udvarok főleg rekreációs célokat szolgálnak a háztömbök, iskolák és kórházak esetén. A COMSOL Multiphysics szoftver CFD (Computational Fluid Dynamics) modul alkalmazásával a légszennyezők terjedése és koncentrációja modellezhető. A legtöbb hasonló tanulmányban alkalmazott időátlagolt sebességprofil mellett a gravitációt és a magassággal csökkenő atmoszférikus nyomást is figyelembe vettük. Továbbá különböző szélsebességet alkalmazva is elvégeztük a szimulációt és vizsgáltuk a légszennyezők koncentrációjának értékét a vizsgált geometria kitüntetett pontjain. Ennek eredményeként megfigyelhető a légszennyezők koncentrációkülönbsége és ezen különbség alakulása, eltérő meteorológiai környezetben. A kutatómunka folytatásaként mérési adatok alapján a modell validálását, majd a már validált modellt eltérő valós, városi helyzetek szimulációját tervezzük elvégezni.

Summary

Nowadays in the urban living environment air quality is one of the most important focus of interest. Urban air quality is affected by many factors. Traffic emission in the streets is a large factor of pollutant concentration. This study focuses on air pollution generated by transport. Can be found many studies in literature about CFD (Computational Fluid Dynamics) modelling in street canyons. However, courtyards, which are the focus of this study are especially important and less studied area of modelling of air pollutant emission. Courtyards are usually used for recreation by residents between blocks of flats, schools, and hospitals. By performing simulations based on dispersion model, concentration of air pollutants could be predictable in these areas as well. Using a simulation software – COMSOL Multiphysics CFD module – dispersion and concentration of air pollutants can be calculated in complex 3D geometries. Beside the commonly used equilibrium boundary layer velocity profile, gravitation and decreasing atmospheric pressure by height were considered in the proposed model. Furthermore, concentrations in specific points of the investigated geometry were calculated in case of different wind speed. As a result, difference of concentration of the investigated air pollutant can be determined in case of different meteorology conditions. Continuing this work the most appropriate simulating method could be chose for modelling real situations using experimental data.

Introduction

Since outdoor air pollution is a major environmental problem in the built environment, number of affected urban inhabitants are increasing mainly caused by transport, industry and heating [1]. Many studies can be found in literature which confirm the positive correlation between high atmospheric concentration of air pollutants and the increasing mortality [2][3]. In 2012 death of around 7 million people were caused by air pollution, which is now the largest environmental health risk in the world - published by WHO. [4]. Traffic emissions in these street canyons are a large contributor to local pollutant concentrations [5]. Additionally, inside of the cities and towns the emissions can be accumulated between buildings, especially where the building density is high which indicates an increase of the pollutants concentration because of reduced airflow [6]. In this point of view courtyards are especially important areas by their role as recreational locations for children playing and people recovering, between blocks of flats, schools and hospitals. In spite of there are courtyards in most of the urban areas, only a few study focuses on investigating of air quality of these areas. Most of the studies focus on modelling temperature conditions, especially in hot climate areas [7].

The modelling methods used for near-field pollutant dispersion is characterized by the complex interaction between the atmospheric flow and the flow around buildings [8]. In the past decades, development of computer technology has contributed the spread of application of CFD (Computational Fluid Dynamics) models in several area for example modelling dispersion of outdoor pollution and accidents [9][10][11]. In the case of urban areas applying CFD models complex geometries could be investigated. In the aspect of modelling dispersion of air pollutants in courtyards, CFD models could be the most representative, which are used for modelling gas dispersion in many cases of street canyons [12][13].

The aim of this study is to determine the concentration of air pollutant component in the simulated courtyard applying different boundary conditions. We analyse how the model complexity affects the pollutant accumulation in the courtyard. In the simplest model a fixed velocity and pressure are considered at the inlet and outlet boundaries. This model complexity is similar to those are usually applied in the modelling of air pollutants spreading in city environment. However, in our work the basic model is extended with velocity and pressure profiles respect to the height and also with gravitational force. The concentration of air pollutant is calculated using all the models to check how the model complexity improve the accuracy of the calculations.

Methodology

Three-dimensional steady state simulations have been performed to study the air quality in a courtyard using CFD module in COMSOL Multiphysics software.

The applied Albebraic yPlus turbulence model is a simplified turbulence model based on the distance to the nearest wall. The local Reynolds number Re = Uy/ν is formed with the local absolute value of the velocity and the distance to the nearest wall. This implicitly assumes that the main flow direction is parallel to the wall. The Algebraic yPlus turbulence model is consistent with a no slip boundary condition, that is u=0. This model is one of the least computationally intensive, but generally the least accurate turbulence model. Hence, Validation of the proposed courtyard model is crucial in our work.

The current analysis is mainly focused on the influence of the small scale geometry. Therefore, the computational domain is a hexahedron with dimensions 50 m by 12 m in the horizontal plane and 40 m in the vertical direction. The computational domain was built using tetrahedral elements with a coarse resolution. The final number of the computational cells used is 60 283.

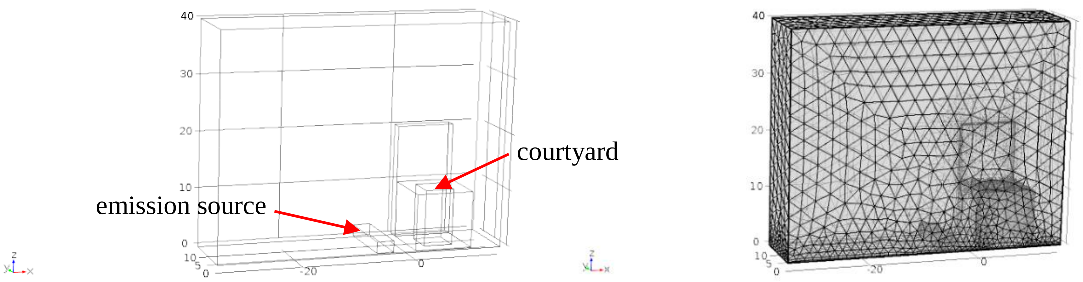


Figure 1. The computational domain and the computational cells (Mesh)

As the inlet boundary condition an equilibrium boundary layer velocity profile (equation (1)) was applied in the simulations.

$$U(z) = \frac{u_*}{\kappa} \ln \frac{z + z_0}{z_0}, \qquad (1)$$

where $U(z)$ is the average wind speed at the height (z) above the ground, $z_0(0.5)$ is the surface roughness, $u_*(0.52)$ is the friction velocity and κ (0.40) is the von Karman's constant. Symmetry boundary conditions are specified on the top and lateral sides of the computational domain. A decreasing pressure by height (101 325– z*10) as an outlet condition was considered as the outflow boundary in the S2 and S4. The influence of gravity on the flow pattern was also considered in the case of S3 and S4: $F_y = -\rho$. Different surface roughness values (z_0) were applied in the last three simulations (Simulations: S5 and S7):

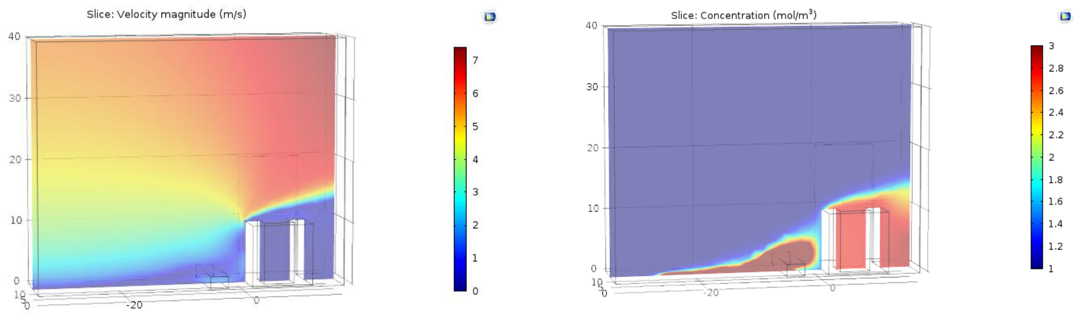
Table 1. Properties of the simulations (S1-S7)

	z_0	**outlet p_0 (Pa)**	**gravity**	**source (1.5 mol/m^3*s)**
S1	0.5	0	-	1.5
S2	0.5	101 325 Pa – z*10	-	1.5
S3	0.5	0	+	1.5
S4	0.5	101 325 Pa – z*10	+	1.5
S5	1.5	101 325 Pa – z*10	+	1.5
S6	2	101 325 Pa – z*10	+	1.5

Results

The results from the simulations are illustrated in simulation images Figure 3. On the left side of the paper the figure represent the velocity values applying only the commonly used wind velocity profile (S1), using decreasing atmospheric pressure by height (S2), considering gravitational force (S3) and applying all these physical condition in one simulation (S4). Concentration values are represented in the right side images.

The outputs of the simulations represent a significant decrease in velocity magnitude from S1 to S4. The concentrations field of the pollutant are related to the velocity profile.



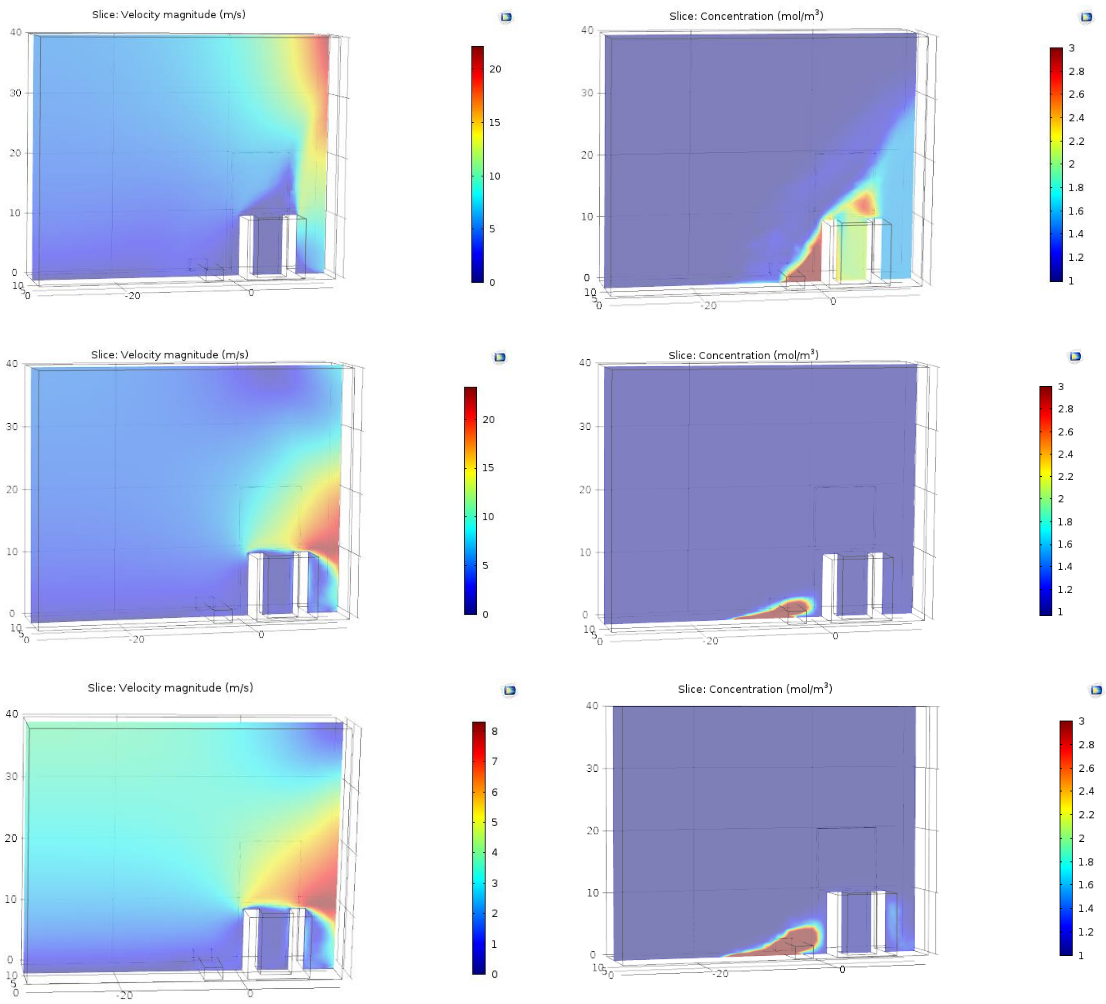


Figure 2. Simulations (S1-S4) of velocity magnitude (left) and concentration of pollutant (right) using different physical conditions

Effect of physical properties could be seen in the simulations (Figure 3.). In specific points of the model tendency of concentration level is similar, but in the S1 concentration is higher in each point followed by concentration values of the S4 and S2. Finally, in the case of S3 could be seen the lowest values of concentration. Difference between values of the commonly used S1 and the most realistic S4 is between 0.72 and 1.76 mol/m^3.

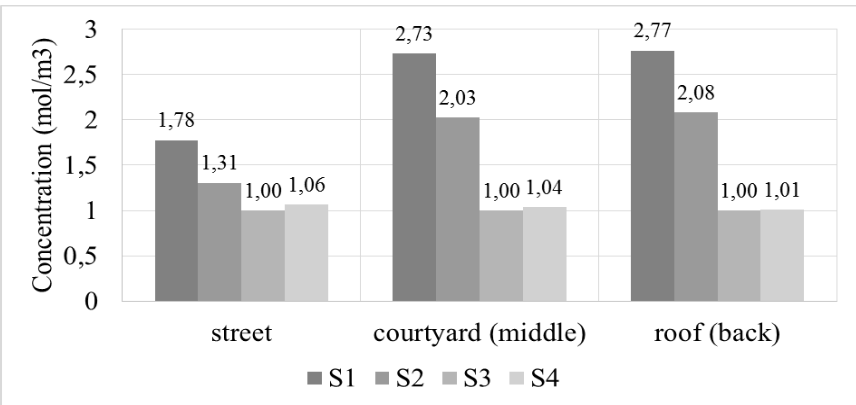


Figure 3. Concentration level in specific points of the simulations (S1-S4)

In the next two simulation (Figure 4.) concentration of gas was simulated by using different roughness values (z_0). Except roughness value physical conditions are the same as in S4. In case of S5 a value (z_0=1.5) and S6 (z_0=2) were considered which are used for simulating velocity profile of a metropolitan area with higher buildings.

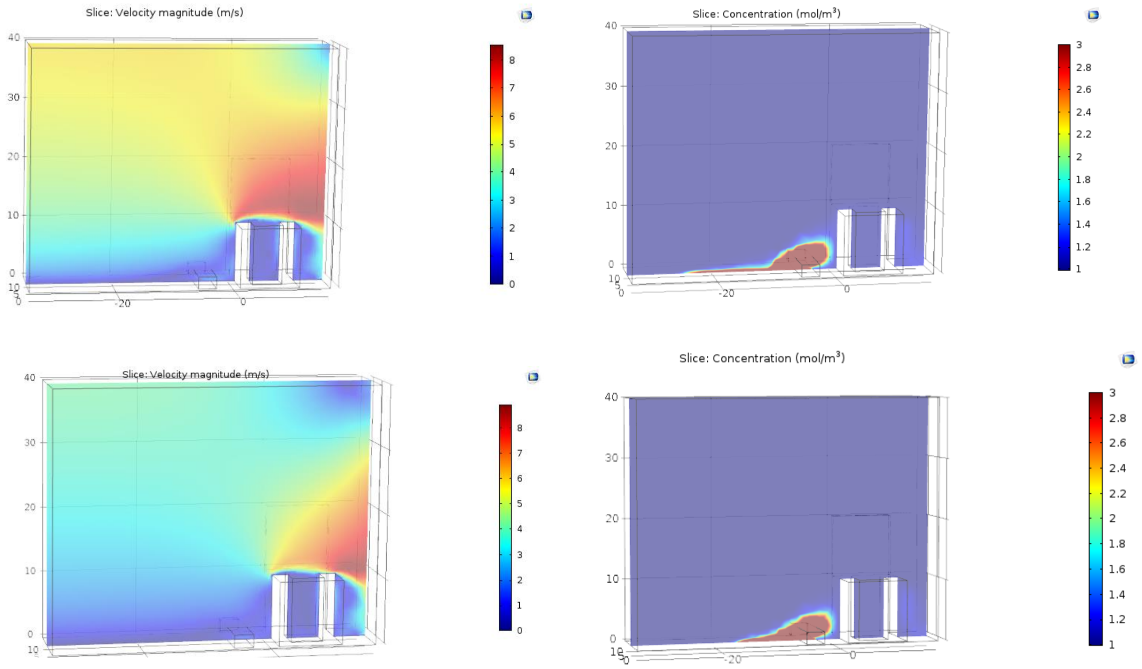


Figure 4. Simulations (S1-S4) of velocity magnitude (left) and concentration of gas (right) using different physical conditions

In Figure 5. differences of pollutant concentrations are represented in a specific point inside the courtyard. In case of S4 the velocity magnitude was the highest, and in case of S6 the lowest at the Inlet side. In spite of that, the lowest velocity magnitude is connected to S6 (z_0=1.5). The expected tendency is represented in concentration values which follow the decreasing values of velocity magnitude at the Inlet side.

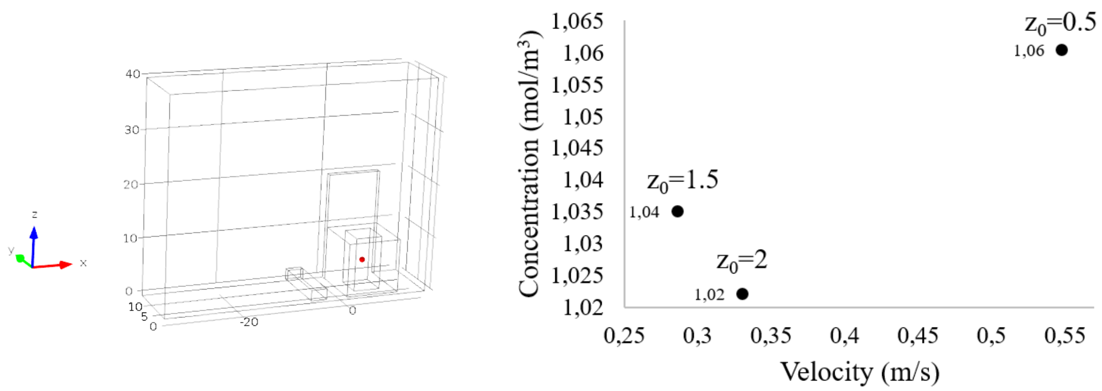


Figure 5. Values of concentration in specific points of the simulations depending on velocity magnitude

Conclusion

Concentration of an air pollutant in the simulated courtyard could be determined in case of applying different boundary conditions. According to six different simulations applying the commonly used velocity profile may not specify the most realistic result. Concentration of pollutant is affected by different physical conditions. Pollutant concentration in a specific point inside the courtyard is strongly affected by wind speed and thus by the applied method. In the future studies applying more different velocity profile should be performed to examine the correlation between wind speed and concentration level. In the following studies the effect of wind direction could be considered. Continuing this work the most appropriate simulating method could be chosen for modelling real situations using experimental data. The accuracy of the computational method is highly depends on the number of computational cells which should be considered in the future studies. The applied CFD simulation methodology is potentially less expensive, less time-consuming than the common used wind tunnel tests.

References

[1] International Energy Agency: World Energy Outlook 2014.
https://www.iea.org/publications/freepublications/publication/WEO2014.pdf

[2] Dehbi, H., Blangiardo, M., Gulliver, J., Fecht, D., Hoogh, K., Al-Kanaani, Z., Tillin, T., Hardy, R., Chaturvedi, N., L Hansell, A. Air pollution and cardiovascular mortality with over 25 years follow-up: A combined analysis of two British cohorts. *Environment International*, Available online (2016)

[3] Chappie, M., Lave, L. The health effects of air pollution: A reanalysis, *Journal of Urban Economics*. 12 (3), 346-376 (1982)

[4] World Health Organization: 7 million premature deaths annually linked to air pollution http://www.who.int/mediacentre/news/releases/2014/air-pollution/en/

[5] Gerdes, F., Olivari, D. Analysis of pollutant dispersion in an urban street canyon. *Journal of Wind Engineering and Industrial Aerodynamics*, 82 (1–3), 105-124 (1999)

[6] Rock, B.A., Moylan, K.A. Placement of ventilation air intakes for improved IAQ. *ASHRAE Trans*. 105 (1), 1-9 (1999)

[7] Ghaffarianhoseini, A., Berardi, U., Ghaffarianhoseini, A. Thermal performance characteristics of unshaded courtyards in hot and humid climates. *Building and Environment*, 87, 154-168 (2015)

[8] Lateb M., Meroney R.N., Yataghene M., Fellouah H., Saleh F., Boufadel M.C. On the use of numerical modelling for near-field pollutant dispersion in urban environments − A review. *Environmental Pollution*, 208 (A), 271-283 (2016)

[9] Gromke, C., Blocken, B., Janssen, W., Merema, B., Hooff, T. van, Timmermans, H. CFD analysis of transpirational cooling by vegetation: Case study for specific meteorological conditions during a heat wave in Arnhem, Netherlands. *Building and Environment*, 83, 11-26 (2015)

[10] Tominaga, Y., Stathopoulos, T. Ten questions concerning modeling of near-field pollutant dispersion in the built environment. *Building and Environment*, 105 (15) 390-402 (2016)

[11] Hanna, S. R., Hansen, O. R., Ichard, M., Strimaitis, D. CFD model simulation of dispersion from chlorine railcar releases in industrial and urban areas. *Atmospheric Environment*, 43 (2), 262-270 (2009)

[12] Vardoulakis, S., Gonzalez-Flesca, N., Fisher, B.E.A. Assessment of traffic-related air pollution in two street canyons in Paris: implications for exposure studies. *Atmospheric Environment*, 36 (6), 1025-1039 (2002)

[13] Vardoulakis, S., Fisher, Bernard E.A., Pericleous, K., Gonzalez-Flesca, N. Modelling air quality in street canyons: a review. *Atmospheric Environment*, 37 (2), 155-182 (2003)