DOI: 10.2428/ecea.2012.19(06)063 ECOL CHEM ENG A. 2012;19(6):619-628

Renata GNATOWSKA[1]

NUMERICAL MODELING OF POLLUTION DISPERSION AROUND THE OBJECTS IN TANDEM ARRANGEMENT

NUMERYCZNE MODELOWANIE ROZPRZESTRZENIANIA SIĘ ZANIECZYSZCZEŃ GAZOWYCH WOKÓŁ OBIEKTÓW W KONFIGURACJI TANDEM

Abstract: The dispersion of pollutants in space around wind engineering structures governed by convection and diffusion mechanism and depends strongly on the velocity field. To understand the phenomena related to the forming of concentration fields it is necessary to recognize the local features of the flow around the objects with the special emphasize for the mean velocity direction, random and periodical fluctuations accompanying the vortex generation in bodies neighbourhood. The specific flow conditions generated around bluff bodies arrangement make it possible to study the gas pollutant dispersion for the case of very complex velocity field typical for built environment.

This paper evaluates the performance of computational modelling approach (RANS k-ε) for two test cases with wind tunnel experiments for validation. The paper presents the results of the complex research program aimed at understanding a character of the flow field in neighborhood of bluff-bodies immersed in a boundary layer and characteristics of pollutants dispersion in that area. The analysis has been performed for the 3D case of two surface-mounted bluff bodies arranged in tandem. The local characteristics of flow and concentration profiles of tracer gas (CO_2) for various inter obstacle gaps were obtained by the use of commercial CFD code (FLUENT). Characteristic feature of flow field around groups of buildings in urban areas is high level the unsteady phenomena resulting from itself character of the wind or from the interference of the wake flow connected with a process of vortex shedding. This is the factor affecting process of the dispersion of pollutants in the built-up area acting more complex the mechanism of propagate of small parts explained on the basis of processes of advection and turbulent diffusion.

Keywords: numerical modeling of pollution dispersion, buildings arrays

The pollutant dispersion in urban areas can have negative consequences on health and comfort of their inhabitants, and on the built environment.. Understanding the transfer and deposition of pollutants in the urban environment is therefore essential in the design process of a building or a ventilation system. Ensuring adequate air quality requires proper aeration of these areas. Its efficiency depends mainly on arrangements of the buildings, the wind direction and locations of emissions sources. The process of

[1] Institute of Thermal Machinery, Technical University of Czestochowa, al. Armii Krajowej 21, 42–201 Częstochowa, Poland, phone: +48 34 325 05 34, email: gnatowska@imc.pcz.czest.pl

pollution dispersion is mainly influenced by mechanisms of mass diffusion, caused by concentration gradients and advection which transfers pollutants in flow direction through mean air movement. Another important factor affecting the entrainment of pollutants into and out of the wake region is the unsteadiness of the wake caused by the shedding of corner vortices. Important role is also played by turbulent transport processes [1, 2]. Improvement in air quality on a local scale and limitation of effect of pollution on human health requires consideration of all the listed factors.

Amongst the methods commonly used to establish better understanding of the wind flow and dispersion processes in the atmospheric boundary layer, *computational fluid dynamics* (CFD) is increasingly explored. Due to the turbulent nature of the wind flow around buildings, the accuracy of the CFD approach applied to urban dispersion problems is to a large extent determined by the performance of the turbulence approach and turbulence model used.

This paper presents the numerical tests of the qualification of the relation between a structure of the flow field in complex urban terrain and characteristics of pollutants dispersion. The *Reynolds Averaged Navier-Stokes* (RANS) turbulence modeling approaches have been used with the k-ε model in RNG version. This choice is made following the work of Baik et al [3] who studied the dispersion of reactive pollutants in an urban street canyon with use the RNG k-ε turbulence model and shows good agreement between the numerical simulation and wind tunnel experiment. The numerical results are compared with experimental data presented in previous article [4]. The aim of the study was to determine the impact of objects configuration, their degree of "immersion" in the boundary layer for the spread of the tracer gas emitted in the vicinity of two cubes in tandem arrangement.

Methods of analysis

Description of the case study

The geometries of the analyzed cases of two obstacles and location of the source in relation to the investigated arrangement of the objects as well as the assumed coordinate system are presented in the Fig. 1. All the measurements were carried out for the

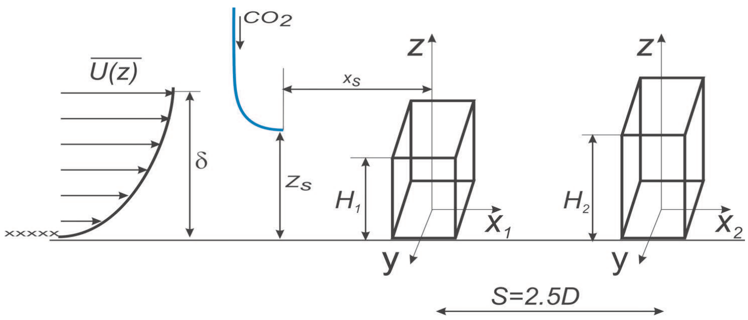

Fig. 1. Schematic diagram of objects in tandem arrangement used in the numerical simulations and nomenclature

Reynolds number $Re_D = 3.4 \cdot 10^4$ based on the free stream velocity $U_\infty = 13$ m/s. The source of emission of carbon dioxide used as a gas marker during the investigations was a pipe located on the symmetry line of the objects at the distance of $x_s = -2.5$ D and at the height $z_s = H_1$ before the windward one. The CO_2 emission rate was assumed $Q = 5\ dm^3/min$.

The results of study presented in this work relate to a fixed ratio of object height $H_1/H_2 = 0.6$ and different "immersion" in boundary layer $H_2/\delta = 0.6$, $H_2/\delta = 1.0$.

RANS with the RNG k-ε model

The program of this work consists of comparison of pollutant concentration profiles (obtained as a result of numerical simulation) with aerodynamic characteristics for building-downwash effect. Three-dimensional RANS simulations have been carried out using a commercial CFD code, FLUENT v.6.3, with the RNG version of a k-ε turbulence. According to the literature [4, 5] this model is widely used for flows in a build environment. For the considered configuration the experimental verification of numerical data has been performed in wind tunnel, which details and results were presented in previous article [4]. When using this model in the present study, the momentum, k and ε transport equations are discretized using a second order upwind scheme. Pressure interpolation is performed with a second order scheme. The SIMPLE algorithm is used for pressure-velocity coupling.

In the CFD simulations presented in this paper, the profiles of longitudinal velocity, k and ε are prescribed at the inlet of the domain (Fig. 2). The oncoming flow profiles have been approximated using the power distribution $U(z) = U_\delta\ (z/\delta)^\alpha$ where α characterizes the terrain type. In addition to the inlet velocity profile, the k-ε RNG

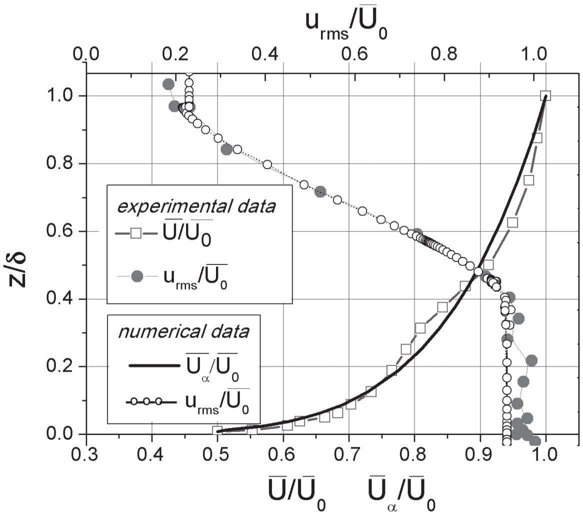

Fig. 2. Distribution of mean velocity U and velocity fluctuations u_{rms} measured in the wind tunnel and implemented as inflow conditions in numerical simulations

turbulence model requires an appropriate distribution of the turbulent kinetic energy k and the dissipation rate ε which together define the velocity and length scale of the turbulent motion. Different definitions of inlet boundary conditions have been analyzed and compared with experimental data [6]. The profiles of U, k and ε that are prescribed at the inlet of the domain are based on the wind tunnel measurements. One of the best methods to describe the inlet boundary conditions is the method proposed by Richards and Hoxey [7]. This, method has been accepted for the calculations described in this paper.

At the outlet, condition "pressure outlet" – provides a uniform distribution of static pressure is prescribed. At the top and lateral boundaries, a symmetry boundary condition is imposed. The domain dimensions have been determined following the COST 732 guidelines (Franke et al, 2007 [8]): 30D (length) · 10D (width) · 10D (height). In the streamwise direction, an empty distance of 6.5D and 20D is allocated upstream and downstream the buildings, respectively. The three-dimensional CFD domain consisted of two objects with a square base (0.04 × 0.04 m) and different heights. The grid independent solution has been obtained for by systematically refining the entire mesh in each direction, increasing the number of nodes by about 50 %. The resolution for tandem arrangement objects has been 298 × 130 × 70. Figure 3 shows the mesh used in the present simulations for a dual objects configuration.

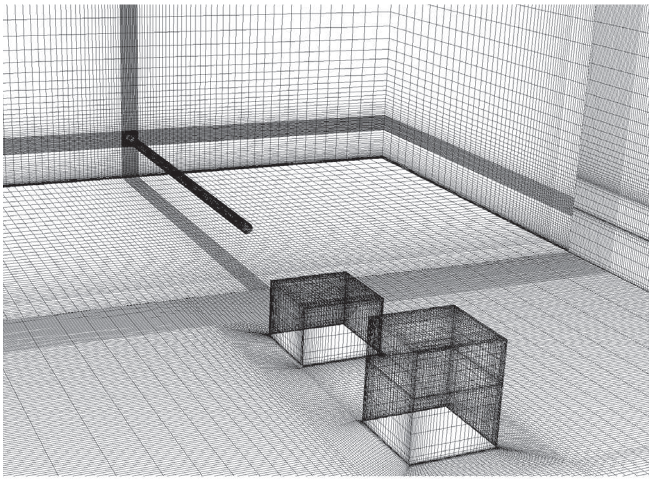

Fig. 3. The computational mesh for objects in tandem arrangement with parameters $H_1/H_2 = 0.6$, $H_2/\delta = 0.6$, $z_S = H_1$

Discussion of the results

The analysis of gas pollution dispersion process requires in-depth identification of the structure of flow around the buildings. The flow structure around three-dimensional bluff-body located on the surface with formed boundary layer is characterized by a high level of complexity.

The case under consideration in this work concerns tandem arrangement which is characterized by H_1/H_2 parameter, which is conducive to occurrence of so-called "downwash" effect $H_1/H_2 = 0.6$. This effect consists in washing of front side of the leeward object with large air masses, which results in strong air circulation in the area between objects, which determines flow structure between them. The zone typical of flow between cuboids are clearly visible in the Fig. 4. The level of modification of flow around the objects of tandem arrangement depends on many factors (distance between elements, change in height of the objects, immersion parameter in boundary layer). In the case of the tandem arrangement, being considered here, in addition with "down-wash" effect the arrangement low-height elements can contribute to the intensification of mixing processes and consequently lead to improved air quality at the zone between objects as it was pointed out by Vanweert and van Rooij [9]. Appropriate design of the wind environment with the presence of emission sources is very important especially taking into account human health and life comfort.

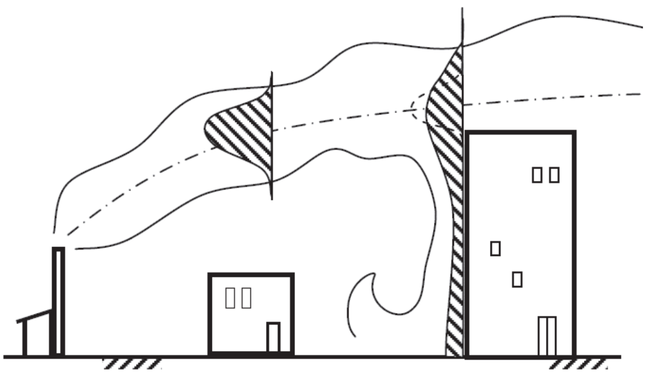

Fig. 4. Scheme of the flow pattern obtained on the base the smoke visualization

The observed modifying impact of interaction between the objects in tandem arrangement is reflected in the results of concentration of the tracer gas emitted in their environment. The Fig. 5 presents the longitudinal component of mean velocity contours distribution in the plane $y/D = 0$. In order to ensure proper comparison of the results, one should reduce the analysed values with the values measured in the place of build-up of the objects. A reference value is the value of velocity registered at the height H_1 in undisturbed flow in the place of object build-up U_{H_1}. Flow around objects with varied height and flow of air from the side of lower element $H_1/H_2 = 0.6$ is conducive to occurrence of downwash effect. This effect is characterized by strong air circulation in the zone between objects, totally determining the structure of flow between each other. Strong air circulation between objects determines rise in velocity of the factor with the direction opposite to the direction of flow of main wind directly adjacent to the ground. No considerable difference in the structure of flow behind the system was observed in the analysed cases. In all the analysed configurations, the air detached on windward wall in first element is detached again in the second object. Renewed detachment of

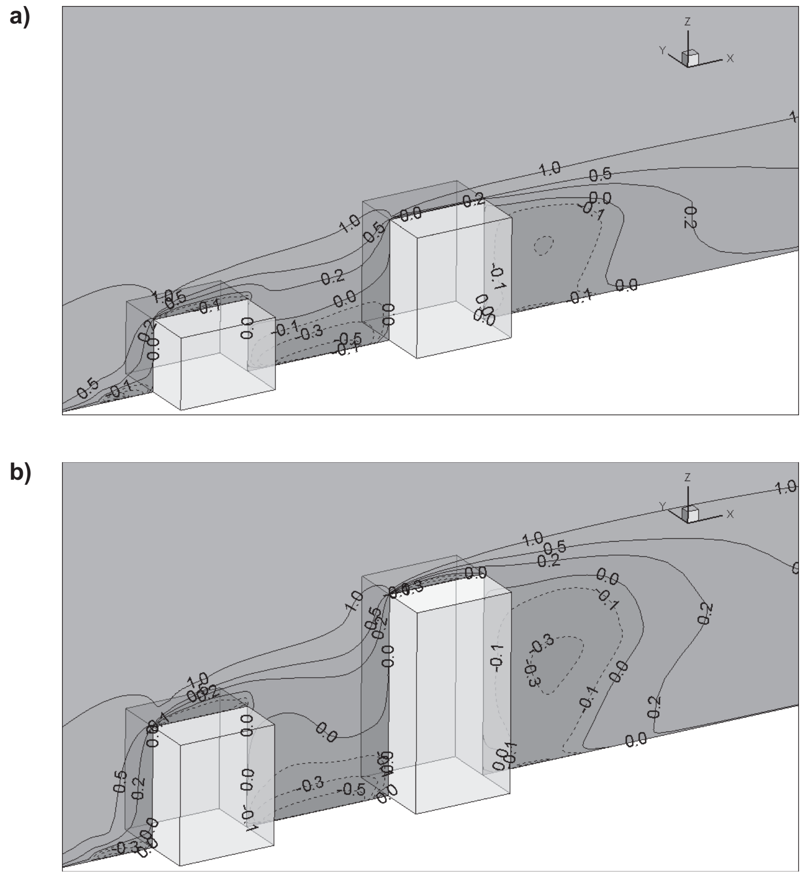

Fig. 5. The contour of longitudinal component of mean velocity U/U_{H_1} in the symmetry plane: a) $H_2/\delta = 0.6$; b) $H_2/\delta = 1.0$

flow from upper surface of the leeward element causes that the length of recirculation zone behind the system remains actually unchanged.

Disturbing impact of the second object on the CO_2 distribution around the analysed arrangement of cubes is illustrated in next figures which shows the distribution of CO_2 concentration in and behind the objects gap for the considered configurations. The general behavior of the concentration fields in numerical and experimental measurements are similar. Fig. 6 and 7 presents cross-sectional distribution of concentration in the gap between objects for both considered configurations: as experimental and numerical results, respectively. The numerical simulations are verified by the experiments performed by Gnatowska and Moryn [4].

It can be seen the high value of the gas marker concentration, emitted at the height of the windward object, at a certain distance from the source and a decrease along the flow direction. The gas marker is moved mainly through upper flow. When the CO_2 plume arrives to front side of the leeward object, which results the strong value of

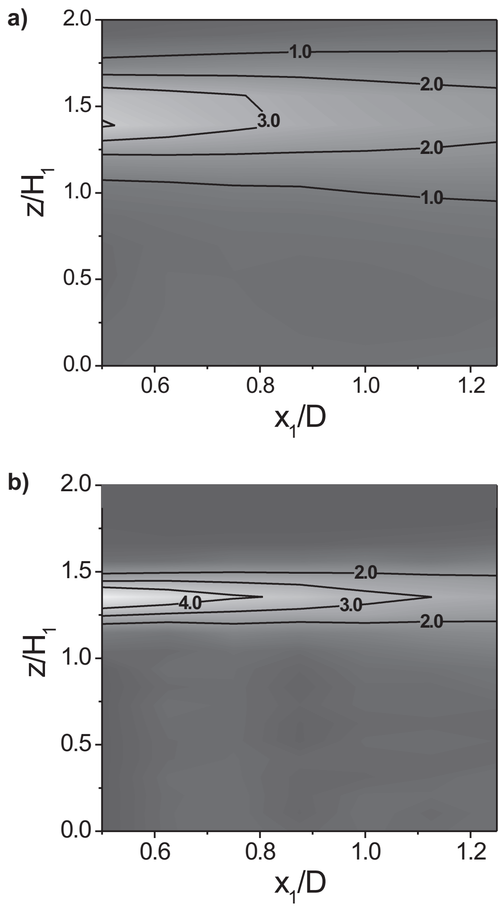

Fig. 6. Experimental measurement – Distribution of mean concentration CO_2 ($z_S = H_1$; $y/D = 0$) in the inter-obstacle gap for different the immersion parameter: a) $H_2/\delta = 0.6$; b) $H_2/\delta = 1$

concentration in that area. The images that we can observed between objects are the result of the flow structure between them. This situation is confirmed in figures, which shows the lower values of pollutant concentration behind the array of two cuboids. It is caused by the recirculation bubble in the gap region between objects in tandem.

Change in height of the elements of the arrangement impacts on changes in the immersion parameter in boundary layer. As results, the biggest changes in flow field are observed in the area between objects. Rise in object height in relation to layer thickness causes rise in impact of windward object and increase in width and length of recirculation zone and extension of the area taken by vortices.

General the numerical results are agreed well with experimental data, but the main observation from all two cases is that the numerical simulations little overestimate the cross-wind (lateral) dispersion of the pollutants. The differences appear practically for each location of the measuring traverse. Further investigation should be directed

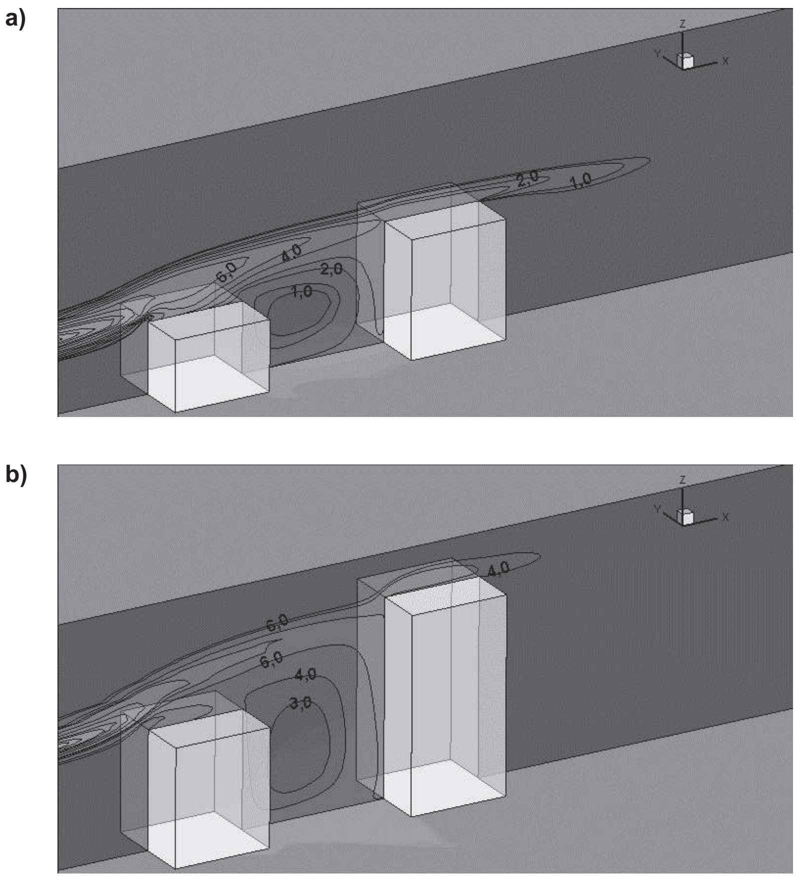

Fig. 7. Numerical simulation – Distribution of CO_2 Mass Fraction [%] ($z_S = H_1$; $y/D = 0$) in the inter-obstacle gap for different the immersion parameter: a) $H_2/\delta = 0.6$; b) $H_2/\delta = 1$

towards identifying the main reasons for numerical modelling discrepancies based on idealized case studies, to support future more complex case studies.

Conclusions

The performed numerical research was aimed primarily at the development of the existing knowledge of the interaction between objects located on the ground and its influence on the pollutant dispersion.

In present experimental study of CO_2 concentration fields around bluff-bodies in tandem arrangement have been observed for case so-called “downwash” effect $H_1/H_2 = 0.6$. The results of the present work showed good agreement between the results of numerical simulation and the wind tunnel experiments for both wind flow and concentration field. Velocity field obtained from the RNG k-ε turbulence model shows the reverse flow in the gap between objects and in the near wake region.

In upper part of first building and between them occurs high level of pollution dispersion, which slowly reduces as the plume moves up the behind objects agreement. The diffusion concentration was very high near the source location. The concentration results with the RNG k-ε model were observed in agreement with the wind tunnel results. The area of maximum concentration is wide, occupying the width of the building.

The numerical simulation results indicate that differences in flow and pollutant dispersion strongly depend on building configuration. A further systematic investigations for various building arrays are needed. Also, a study is required to explain reasons for the variations in flow and pollutant dispersion.

Acknowledgements

The paper was supported by the State Committee for Scientific Research under Project No. BW 1-103-204/08 titled "Modeling of wind environment as part of urban planning".

References

[1] Blocken B, Stathopoulos T, Saathoff P, Wang X. J Wind Eng Ind Aerodyn. 2008;96(10-11):1817-1831. DOI: 10.1016/j.jweia.2008.02.049.
[2] Moryn-Kucharczyk E, Gnatowska R. Pollutant dispersion in flow around bluff bodies arrangement. In: Wind Energy. Peinke J, Schaumann P, Barth S, editors. Proc Euromech Colloquium. Berlin, Heidelberg: Springer-Verlag; 2007.
[3] Baik JJ, Kang YS, Kim JJ. Atmos Environ. 2007;41:934-949. DOI: 10.1016/j.atmosenv.2006.09.018.
[4] Gnatowska R, Moryn-Kucharczyk E. Proc ECOpole. 2010;4(1):37-42.
[5] Ferreira AD, Sousa ACM, Viegas DX. Prediction of building interference effects on pedestrian level comfort, J Wind Eng Ind Aerodyn. 2002;90:305-319. DOI: 10.1016/S0167-6105(01)00212-4.
[6] Gnatowska R. Numerical modeling of unsteady phenomena in flow around bluff-bodies in tandem arrangement. Proc 3rd International Conference on Experiments /Process/ System Modelling/ Simulation/ Optimization (IC-EpsMsO), Athens. 2009:485-491.
[7] Richards PJ, Hoxey RP. Appropriate boundary conditions for computational wind engineering models using the k-e turbulence model. J Wind Eng Ind Aerodyn. 1993;46-47:145-153. DOI: 10.1016/0167-6105(93)90124-7
[8] Franke J., Hellsten A., Schlünzen K.H., Carissimo B. The COST 732 best practice guideline for CFD simulation of flows in the urban environment – A summary. Int. Journ. of Env. and Poll, 2011;44:419-427.
[9] Vanweert FLH, van Rooij JIJH. Air-quality and spatial planning, Proc PHYSMOD 2007:191-196.

NUMERYCZNE MODELOWANIE ROZPRZESTRZENIANIA SIĘ ZANIECZYSZCZEŃ GAZOWYCH WOKÓŁ OBIEKTÓW W KONFIGURACJI TANDEM

Instytut Maszyn Cieplnych
Politechnika Częstochowska

Abstrakt: Rozprzestrzenianie się zanieczyszczeń w przyziemnej warstwie atmosferycznej regulowane jest przez mechanizm dyfuzji i konwekcji oraz silnie zależy od pola prędkości. W celu zrozumienia zjawisk związanych z kształtowaniem się pól koncentracji istotne jest rozpoznanie struktury przepływu wokół obiektów ze szczególnym uwzględnieniem prędkości średniej oraz jej losowej i okresowej składowej towarzyszącym generacji wirów w otoczeniu obiektów naziemnych w obszarach zabudowanych.

W pracy przedstawiono wyniki numerycznego modelowania procesu dyspersji zanieczyszczeń gazowych w strefie zabudowanej. Ich celem było określenie wpływu konfiguracji obiektów, stopnia ich „zagłębienia” w warstwie przyziemnej, a także położenia źródła emisji na rozprzestrzenianie się zanieczyszczeń (znacznik gazowy – CO_2). Badany układ stanowiły dwa trójwymiarowe modele budynków o różnych wysokościach ustawione w tandemie. Charakterystyki aerodynamiczne przepływu oraz profile koncentracji gazu znacznikowego (CO_2) dla różnych konfiguracji obiektów uzyskano z wykorzystaniem komercyjnego oprogramowania FLUENT. Cechą szczególną pól prędkości w otoczeniu grupy opływanych budynków jest wysoki poziom niestacjonarności wynikający zarówno z samego charakteru wiatru, jak i z faktu generowania przez obiekty zjawisk periodycznych związanych z procesem schodzenia wirów. Jest to czynnik, który oddziałuje na proces dyspersji zanieczyszczeń w obszarze zabudowanym, czyniąc jeszcze bardziej złożonym mechanizm rozprzestrzeniania się cząstek tłumaczony na bazie procesów adwekcji i turbulentnej dyfuzji.

Słowa kluczowe: numeryczne modelowanie dyspersji zanieczyszczeń, układy budynków