Proceedings of ECOpole
Vol. 5, No. 1 2011

Renata GNATOWSKA[1]

NUMERICAL MODELING OF POLLUTANT DISPERSION IN FLOW 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. 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 modelling of pollutant dispersion, buildings arrays

The analysis of pollutant dispersion throughout atmospheric boundary layer has grown in importance since human activity has become so intense that it started having considerable impact on natural environment. The level of concentration of pollutants has grew, particularly in urban areas and it impacts on their inhabitants.

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 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.

With a steady growth in computer technology, the *computational fluid dynamics* (CFD) has emerged as an effective tool to establish better understanding of the wind flow and dispersion processes that occur in the atmospheric boundary layer. The study of pollution entrainment in urban areas with different arrangements of bluff bodies is a new stem of environmental engineering. Since the analysis of flow field in complex geometries

[1] Institute of Thermal Machinery, Czestochowa University of Technology, al. Armii Krajowej 21, 42-200 Częstochowa, phone 34 325 05 34, fax 34 325 05 07, email: gnatowska@imc.pcz.czest.pl

is very costly, the simple pollution dispersion around a cubes mounted on a surface is still under investigation.

This paper presents the numerical tests of the qualification of the relation between a structure of the flow field in complex urban terrain (in the built-up area) and characteristics of pollutants dispersion. The numerical results are compared with experimental data presented in previous article [3]. The aim of this work 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 rectangular blocks in tandem arrangement.

Methods of analysis

The program of this work consists of comparison of pollutant concentration profiles with aerodynamic characteristics for building-downwash effect (obtained as a result of numerical simulation).

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 Figure 1. The three-dimensional CFD domain consisted of two objects with a square base (0.04 x 0.04 m) and different heights. The total height, width and length of the domain was accordingly 10 x 10D and 30D. The inflow boundary layer has been prescribed according to the power law distribution $U(z) = U_\delta(z/\delta)^\alpha$, where α characterizes the terrain type.

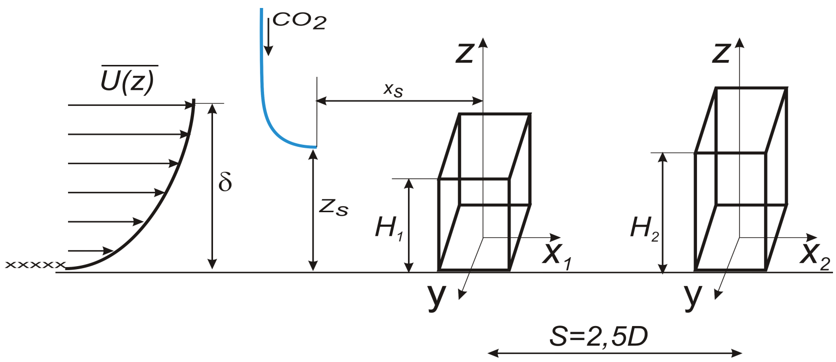

Fig. 1. Schematic diagram of objects in tandem arrangement used in the numerical simulations and nomenclature

All the measurements were carried out for the 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 $x_s = -2.5D$ 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 their "immersion" in boundary layer $H_2/\delta = 0.6$.

Three-dimensional RANS simulations have been carried out using a commercial CFD code, FLUENT v.6.2, 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 [3].

Discussion of the results

The analysis of gas pollutants 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 image (see Fig. 2a) being the result of smoke visualization The level of modification of flow around the analysed arrangement of objects of tandem type depends on many factors (distance between elements, change in height of the objects, immersion parameter in boundary layer). Figure 2b presents distribution of concentration field in flow around the elements for the source located at the height of the windward object. The results show that the biggest changes in flow field are observed in the area between objects.

The smoke vortex flow visualization (experimental measurements) and concentration contour plots (numerical calculations) are clearly reflected the qualitative nature of the flow in the area between the buildings (see Fig. 2ab). 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.

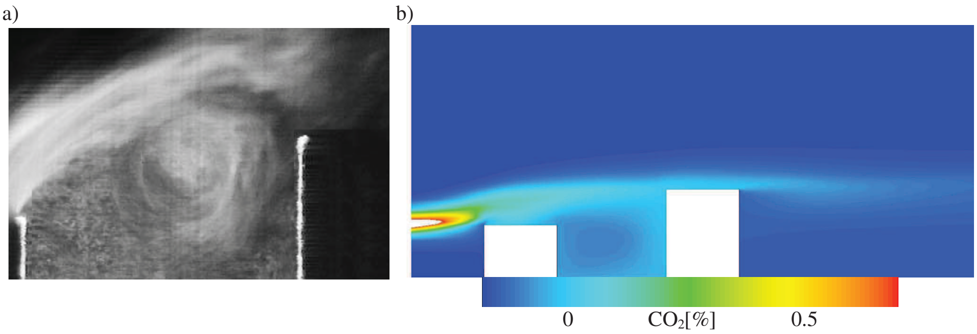

Fig. 2. The flow patterns obtained with smoke visualization technique - results of wind-tunnel experiments (a), The CO_2 concentration contour plots at plane y/D = 0 for considered configuration objects in tandem arrangement - results of numerical calculations (b)

In the case of the tandem arrangement, being considered here, in addition with "downwash" 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 & van Rooij [6]. Appropriate design of the wind environment with the presence of emission sources is very important especially taking into account human health and life comfort.

Disturbing impact of the second object on the flow around the analysed arrangement of cubes causes many changes. This is illustrated by Figure 3 which shows the cross-sectional distribution of CO_2 concentration in and behind the objects gap for the considered configuration. The general behavior of the concentration field is similar to obtain in experimental measurement. 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. The pollutant plume width predicted at the pedestrian level and at the ground level in gap between objects is wider than in other part of gap flow. When the CO_2 plume arrives to front side of the leeward object, which results the strong value of 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 Figure 3 (see $x_2/D = 0.5$), 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.

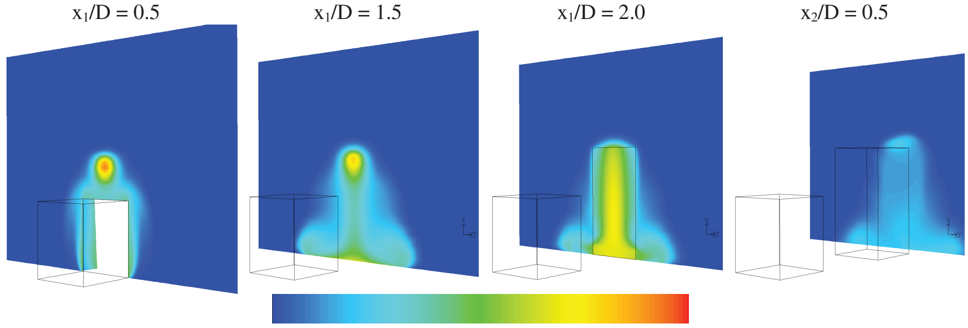

Fig. 3. The CO_2 concentration contour plots at various planes x/D = const for considered configuration objects in tandem arrangement - results of numerical calculations

The differences appear practically for each location of the source and each measuring traverse. The highest values of marker gas in tandem arrangement axis ($y/D = 0$) are observed for $z_S = H_1$ in the area above the height of windward building, while for the source located on the base, along almost the whole height of the first element. For $z_S = H_1$ the values of gas marker concentration are almost ten times bigger for $y/D = 0$ than $y/D = 0.5$ while for $z_S = 0$ the situation is opposite (twice the difference). In the case of the measurements along the outside edges ($y/D = 0.5$), maximal C_{CO2} values appear in the base.

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. Such studies may contribute to the better understanding of physical processes and provide necessary information for the development of numerical modeling.

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 main attention of this paper 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 rectangular cubes in tandem arrangement.

The presented results show how important for ensuring adequate air quality, human health and their wind comfort, proper formation of wind-related environment of ground objects is.

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. and Wang X.: *Numerical evaluation of pollutant dispersion in the built environment: Comparisons between models and experiments*. J. Wind Eng. Ind. Aerodynam., 2008, **96**(10-11), 1817-1831.
[2] Moryn-Kucharczyk E. and Gnatowska R.: *Pollutant dispersion in flow around bluff bodies arrangement*. Wind Energy, Springer Verlag, Berlin, Heidelberg 2007, 49-53.
[3] Gnatowska R. and Moryn-Kucharczyk E.: *Analysis of pollutant dispersion in flow around the array of buildings*. Proc. ECOpole, 2010, **4**(1), 37-42.
[4] Ferreira A.D., Sousa A.C.M. and Viegas D.X.: *Prediction of building interference effects on pedestrian level comfort*. J. Wind Eng. Ind. Aerodyn., 2002, **90**, 305–319.
[5] Martinuzzi R.J. and Havel B.: *Turbulent flow around two interfering surface-mounted cubic obstacles in tandem arrangement*. J. Fluid Eng., 2000, **122**, 24-31.
[6] Vanweert F.L.H. and van Rooij J.I.J.H.: *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 mechanizmy 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 „zanurzenia" 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 za pomocą procesów adwekcji i turbulentnej dyfuzji.

Słowa kluczowe: numeryczne modelowanie dyspersji zanieczyszczeń, układy budynków