BULLETIN OF THE POLISH ACADEMY OF SCIENCES
TECHNICAL SCIENCES, Vol. 63, No. 3, 2015
DOI: 10.1515/bpasts-2015-0084

Comparative analysis of numerical and experimental studies of the airflow around the sample of urban development

K. GUMOWSKI[1], O. OLSZEWSKI[1], M. POĆWIERZ[1], and K. ZIELONKO-JUNG[2]*

[1] Institute of Aeronautics and Applied Mechanics, Faculty of Power and Aeronautical Engineering, Warsaw University of Technology, 24 Nowowiejska St., 00-665 Warsaw, Poland

[2] Department of Structure, Construction and Technical Infrastructure, Faculty of Architecture, Warsaw University of Technology, 55 Koszykowa St., 00-659 Warsaw, Poland

Abstract. In this paper, on the background of a short overview of the recent advances in the field of Environmental Wind Engineering (EWE), a comparison of wind tunnel experiment and numerical simulation for some cases of airflow around an urban layout have been reported. The purpose of the study is quantitative and qualitative comparison of measurements in the wind tunnel as well as numerical simulation using Ansys Fluent software. The study is concerned mostly with the analysis of parameters which are essential in the building industry, such as pressure and velocity fields. In the numerical analysis the *k-ε realizable* model of turbulence with the basic model of boundary layer – *Standard Wall Treatment*, were used. Particular attention has been paid to accurate depiction of the conditions on the inlet and the selection of suitable computing.

Key words: environmental wind engineering, airflow around buildings, CFD simulation, the *k-ε realizable* model of turbulence.

1. Introduction

The progress of the architecture and urbanism raises problems requiring interdisciplinary research conducted with the use of advanced experimental techniques and computer simulation. The fields of this research include aerodynamics, especially in intensively developed area of Environmental Wind Engineering (EWE). This branch allows to explore and consequently to optimize airflow around buildings and urban structures [1, 2]. Microclimate phenomena associated with the airflow around buildings have become of particular interest in the recent years [3–6]. Nowadays quality requirements concerning microclimate conditions (proper ventilation, elimination of rapid air movement in pedestrian area, protection against noise and cooling caused by the wind, reduction of deposition odors and dirt accumulation) [7] imply the need for aerodynamic testing in the initial phase of design process, where decisions concerning shapes of the objects are made.

Initially, experimental studies in wind tunnels which allow visualization and measurement of selected parameters, were used. Wind tunnel research is costly and time-consuming, hence the number of design variants examined is limited. This in turn provides a major barrier for investors. As the result, such experiments are undertaken mainly in the case of buildings with particular aerodynamic conditions (e.g. high-rise buildings, free-standing roofs of large-volume, building assemblies localized within the aeration channels) [8].

In effect of the progress in computer simulations and increasing availability of Computer Fluid Dynamics (CFD) made it possible to use numerical simulation in EWE since the late nineties [3, 9, 10]. Research methods of numerical simulation allow to analyze many variants of building shape and position in short periods of time. Although some recommendations and practice guidelines for such studies are available in literature [11, 12], they require high expertise and a creative approach to each facility and pose a scientific challenge [2–4].

The main drawback of numerical methods is the quality of the obtained results. While the tunnel tests lead to results of a well-estimated accuracy, the quality of the results given by numerical simulation methods depends on the errors in the modeling of physics. Often the geometry and inlet conditions are simplified. Hence numerical models should be developed individually for each object, and the verification of each model by means of comparison with experimental data obtained in a wind tunnel is indispensable (validation process can be done for only two or three wind directions, not for all). The verification of numerical models is especially important in the case of complex sets of buildings, the examples of which include urban development reported in the paper.

The research results of the simple geometric objects are reported in literature. Such examples as: freestanding solids [13–16], regular alignment of solids [3, 17], the space between the two walls showing the street model [18], are often referred. However publications concerning more complex geometric structures, which take into account the surrounding development and as such are closer to the real problems met in architectural design, are relatively rare [5, 19, 20]. Reports on complex assemblies for urban development, containing experimental verification of numerical models are scarce.

The aim of the current study are qualitative and quantitative comparisons of two methods for testing the airflow

*e-mail: katarzyna.zielonko-jung@arch.pw.edu.pl

around urban structures. It comprises of the measurements performed on a real building model in a wind tunnel and numerical simulations. The subject of the research described is the complex set of buildings, which is a hypothetical example of an urban quarter. It represents a typical situation where a C-shaped building is surrounded by a dense city structure.

In the following paper the k-ε *realizable* model of turbulence, recommended and used in several previous works [2, 4, 7, 21, 22], has been applied in the numerical simulations. A proper accuracy of the numerical model and simulation was obtained by selection of an appropriate mesh and a faithful reproduction of flow features at the inlet (obtaining profiles of velocity and turbulence intensities as closely as possible). The wind experiments were performed for several different velocity profiles and turbulence intensities, via accurate measurements of the velocity profile in front of buildings and pressure measurements in the vicinity of buildings along a predetermined path.

2. Models of test object

2.1. Geometrical model. An urban arrangement in the form of an urban quarter filling a rectangular grid of streets was chosen as an object of this research. The model illustrates the spatial structure of a compact city layout in a form of a quarter shown in Fig. 1. It contains two basic types of urban interiors, that is: streets and a square in the shape of a courtyard inside the quarter. There are some simple buildings of an elongated shape and one in the shape of letter C. It is the type of buildings where potentially many diverse phenomena may occur at the same time, for instance, a corner effect, a narrowed flow effect, an elongated building effect, zones of intense turbulence and stagnation zones. Dimensions and proportions of the building layout adopted for the study correspond to the rules of positioning of buildings. These rules take into account insolation premises, mutual shadowing, fire regulations as well as residential and service functionality.

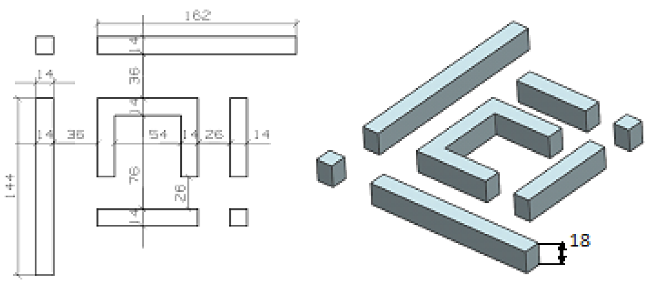

Fig. 1. Shape of the analyzed urban quarter (real dimensions in meters)

The model of a residential quarter including adjacent buildings was made in scale 1/400. The scale was chosen in such a way as to enable a direct comparison between numerical results and the result achieved in the wind tunnel tests.

A section of the wind tunnel with a height of 0.625 m, a width of 1.25 m and length of 2.5 m is the computational domain. According to the recommendation contained in [16], the minimum height of the area should be at least 5-times equal to the maximum height of the building (0.045 m). In case of this study, the ratio is more than twice as high. The velocity inlet is within 0.4 m or about 8 heights of the building, which meets the recommendations given in [9]. The outflow was placed at a distance of 1.5 m from disturbing elements (it is about 30 height of the building).

2.2. Numerical model. Numerical calculations were carried out in the Ansys Fluent software [23, 24]. A model of turbulence was k-ε *realizable* [22], which is an extension of the standard k-ε model [5, 7, 8, 14]. It belongs to the group of *RANS* methods in which the motion of the fluid is described by the equation based on the parameters of the time-averaged flow rate.

According to the Reynolds' hypothesis, intensive quantities (for instance velocity or pressure) in the turbulent motion can be represented as a sum of the mean value and fluctuating variable. The values defined in this way are introduced into continuity and momentum equations. The system which results from this procedure consists of four equations. There is a new element called the Reynolds' stress tensor in the momentum equations. It has six unknown components. The problem is not closed. There are ten unknown variables that is pressure, three components of velocity and six of stress tensor. It is therefore necessary to include the equations which allow to calculate those additional unknowns. All turbulence models containing the above mentioned standard k-ε and k-ε *realizable* deal with this problem [12, 23].

The k-ε models of turbulence are based on the Boussinesq's hypothesis, which assumes that the Reynolds' stresses are proportional to a deformation rate. It is a two equation model. That is due to the fact that turbulent viscosity is defined by two additional differential equations describing the changes of the kinetic energy of turbulence k and the dissipation of kinetic energy ε over time.

The model k-ε *realizable* is a refinement of the basic k-ε model. The main difference between the two models comprise their constant value, which occurs in the definition of turbulent viscosity in the standard k-ε model. In the k-ε *realizable* it becomes a variable determined on the basis of such factors as kinetic energy of turbulence, dissipation of turbulence and the strain tensor. This alternation prevents the formation of negative turbulent normal stresses in fluid, which may occur in the standard k-ε model. In all cases examined so far, the results obtained with this model are closer to reality than those obtained from calculations using the standard model [23, 24]. Therefore, the numerical calculations of flow around the buildings made use of the k-ε *realizable* model.

Turbulence models k-ε and k-ε *realizable* are intended to describe the motion of a high Reynolds' number, and for this class of flows they give the best results [22, 23]. This is because the model k-ε assumes that the viscous stress (depending on the molecular viscosity) is small in comparison to the Reynolds' stress [22]. In the direct neighborhood of walls, however, these assumptions are not met. The Reynolds' number decreases as it approaches the wall, and the proportion of the viscous stress is large because of the relatively large velocity gradients. Therefore, the k-ε model in the vicinity of walls does not give correct results and has to be replaced by a boundary layer model. To determine the flow in the vicinity of walls the *Standard Wall Function* model was applied with respect to [23, 24]. Its use does not require excessively thick mesh around the walls, which allows a great reduction in computation time.

The flow was modeled as stationary, which is a common practice in numerical simulations of flow around buildings.

3. Wind tunnel experiments

The aim of the study was to produce a velocity profile which is the boundary condition for the numerical calculations, as well as providing data for qualitative and quantitative comparison of a numerical simulation and experiment. The measurements were performed in a wind tunnel of an open circuit with a closed test section 1 m × 1 m and 7 m in length.

3.1. Velocity measurement, turbulence intensity and the pressure field. The appropriate velocity profile and turbulence kinetic energy distribution were obtained by inserting an obstacle into the tunnel in order to disturb the flow and achieve an appropriate roughness of the ground level. These obstacles included spires and cubic blocks placed at the bottom of the tunnel in the formation section, in the area before the model. The method used for distribution of these elements to make them suitably disrupt the flow of the wind tunnel was adopted according to the study [25].

Velocity measurements were made both in front of a quarter of buildings (to provide input to numerical analysis) and in the immediate vicinity of buildings (in order to provide data for benchmarking). The velocity profiles were measured by means of a hot wire (probe P11, wire diameter 5 μm) {Dantec Dynamic CTA} in 13 locations along the normal path to the floor. Their location is shown in Fig. 2.

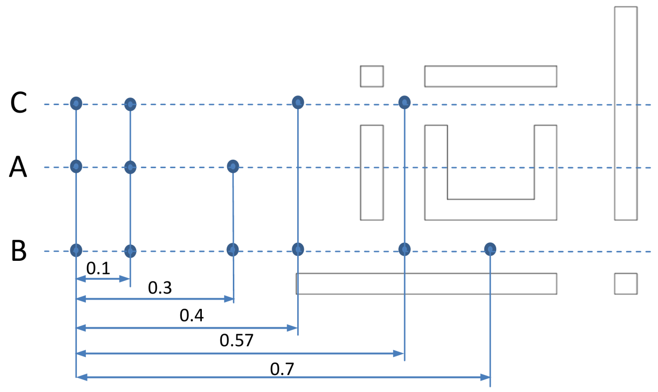

Fig. 2. Positions of measurements of the velocity profile and the intensity of turbulence (XY plane, dimensions of the model in meters)

In each place, velocity and turbulence intensity were measured at approximately 90 points. The highest measurement point was located at a height of about 350 mm above the base of the tunnel. Together with lowering of the probe (and increase of speed gradients) the distance between successive measurement points of logarithmic densification (the smallest steps as 0.05 mm) was reduced. Values that were obtained directly from the measurements included the average speed , and speed fluctuations (standard deviation of the speed) and were measured along the direction of the flow.

Pressure measurements were performed at the floor of the tunnel, near the central building, and ran along designated paths. Location of the holes to measure the pressure on the model is shown in Fig. 3. Accuracy of the measurement is within ± 1 Pa.

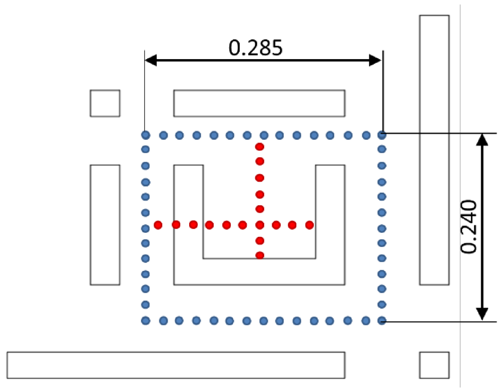

Fig. 3. Positions of pressure sensors (XY plane, dimensions of the model in meters)

3.2. Visualization of flow phenomena using oil film. The method of the oil film belongs to the visualization techniques in which the tested surface is coated with a thin layer of oil containing a suitable pigment. During the test, oil distributed on the surface is blown off from the areas of high flow velocity and it is accumulates in places where the flow is much less dynamic. In addition, the washed away oil leaves more pigment where the flow is slower and the shear stress – less intense. The resulting images provide information about the time averaged velocity (dependent on shear stress) and the streamline [3]. During the experiments pictures were taken every 30 seconds. Their sequence allows for a better understanding and correct interpretation of the phenomena occurring in the flow.

4. Velocity profile and computational grid

From the point of view of the quality of the results an appropriate choice of the distance between the inlet (where the boundary condition is given) and the wall of the first building is extremely important. On the one hand, an extremely long distance can change the shape of the velocity profile in front of an obstacle and may result in discrepancies between calculations and experiment. On the other hand, due to the forced boundary conditions (the derivative of the flow parameters along the normal to the surface of the inlet is zero), the inlet should be sufficiently separated from the object. Disturbances caused by the presence of the model should not influence the adopted inlet data.

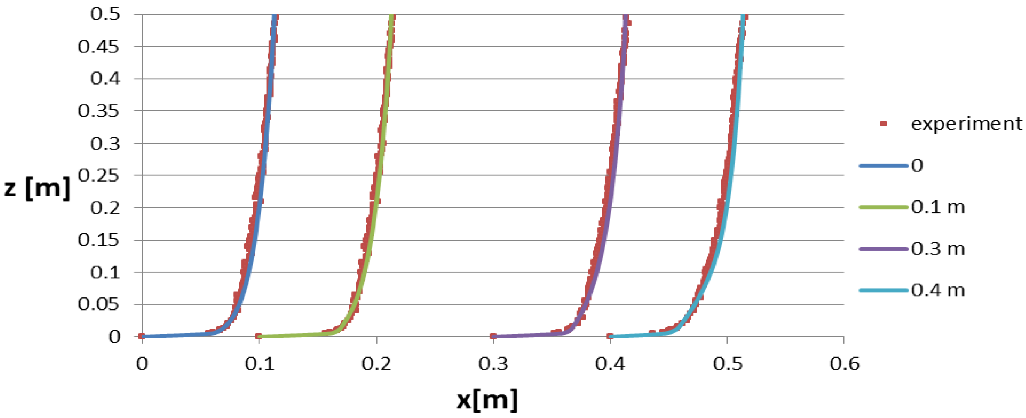

Fig. 4. Development of velocity profiles along a tunnel for four different x values; comparison graphs of velocity used in numerical simulation (colored lines) with experiment measurements (red points)

For the right selection of the distance x between the inlet and the model of buildings, several computational models were tested prior to the start of research for $x = 0.2 - 0.7$ m. It is recommended in the literature that the distance x should be about 5 to 8 height of the tallest building h_b [10]. Following this practice, the main part of the research was performed for $x = 0.4$ m, which is equal approximately 8 h_b.

In typical flows around buildings, an atmospheric boundary layer retains the horizontal homogeneity. This means that the velocity profile does not change substantially along the flow direction and the velocity gradients are close to 0. Limitations turbulence models RANS (in particular, the k-ε and the ones related) have the effect that when using the experimental conditions at the inlet, the velocity profiles have a tendency to change in the numerical simulation of flow along the direction [15, 16]. To reduce this problem, investigations of how the velocity profile develops were done along the length of the tunnel before the profile reaches the front of buildings (as in [16]). The aim of this study was to provide a horizontal homogeneity of the flow in numerical analysis and obtain velocity profiles consistent with the results of measurements in the wind tunnel. The evolution of the velocity profile is shown in Fig. 4, where from the same scale can be read out values of flow velocity in [m/s] units.

Generation of a correct computational grid is extremely important from the point of view of the quality of numerical solution obtained in the study. The construction of the grid was guided mainly by the principles available in the literature [22–24].

The discretization of calculation is shown in Fig. 5. For the purpose of our calculations, a structural grid of 1.8 million items was used. The length of the elements in the grid thickened zone is equal 1/9 the height of the building, which is about 0.005 m. Outside this zone, the elements increase in each direction with growth rate about 1.2.

Two alternative methods of discretization were examined: a) a given grid was concentrated, b) unstructured mesh was used. Verification of the grid's correctness was achieved by comparing the data on pressure from the numerical simulation and from the experiment and by a comparison the dimensionless distance from the wall y^* defined by Eq. (1) in the pressure measurement points

$$y^* = \frac{\rho c_\mu^{1/4} k^{1/2} y}{\mu}, \tag{1}$$

where ρ is a density, $c_\mu = 0.09$ – constant, k – a kinetic energy of turbulence and μ – a turbulence viscosity.

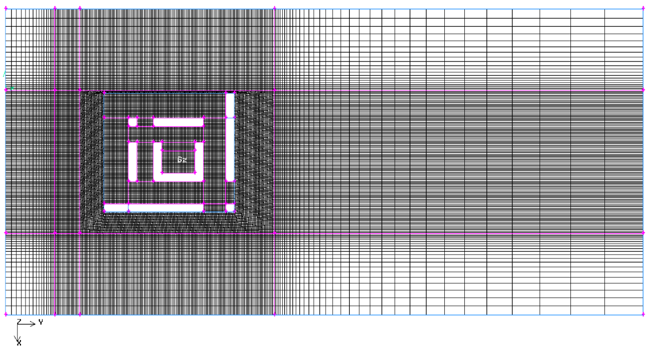

Fig. 5. The mesh in XY plane

The volume of the mesh elements of the walls should be of such a size, that in the middle of them, the dimensionless distance from the wall y^* should be contained within $15 < y^* <\sim 300$ wherein the most desirable is $y^* \in (30, 60)$ [23, 24].

The above analysis showed that: the output mesh fulfills the requirements for dimensionless distance from the wall. All elements in the vicinity of the walls meet the criterion $15 < y^* <\sim 100$ wherein for most of them $y^* \in (30, 60)$. Pressure values obtained by the use of structural grids are closer to the experimental results than the results obtained using unstructured mesh. Mesh refinement does not significantly affect the quality of the results and the time allocated for computations with the use of an output grid is the shortest.

5. Comparison of numerical calculations and experimental results

The results of tunnel experiments and numerical simulations are shown in Figs. 6–14. The research was performed for two opposite directions of flow, along the longer axis of the quarter, marked as case 1 and case 2. Comparative analysis is divided into two parts:

- qualitative – which juxtaposes the oil visualization paintings and the current lines against the background of the velocity field obtained numerically (Fig. 6),
- quantitative, in which the pressure distributions are compared along specified paths (Figs. 8–10, 12–14).

The results of numerical simulations are showed in the plane $z = 2.5$ mm in model dimensions, which is equal 1 m in real dimensions.

From comparison of left and right part of Fig. 6, it can be observed, that the flow patterns obtained by calculation are generally similar to the results of the oil visualization for both presented directions of flow (case 1 and case 2). In most of the test areas, the direction of the flow coincides with the experiment. However, differences are noticed between the results of oil visualization and image of streamlines obtained numerically. They can be divided into three groups:

a) inaccuracies in the mapping of "symmetric" vortices in the immediate vicinity of the narrow buildings,
b) underestimation of the size of the stagnation zone in front of the building – in places where the flow was inverted,
c) incorrect direction of the streamline in the forehead part of stagnation area.

In the case of images obtained by both methods certain aerodynamic phenomena occurring around buildings can be identified, e.g. a corner effect near the gable walls, a narrowed flow effect at the entrance to the street, a stagnation zone inside the quarter. However, these phenomena are more evident in the numerical visualization, as this method contains information approximating the value of tested parameter (in this case the wind velocity). This information is important from the point of view of the use of numerical simulation in concept design phase, when a quick comparison between several variants of building design and general understanding the consequences that may follow spatial decision-making are crucial.

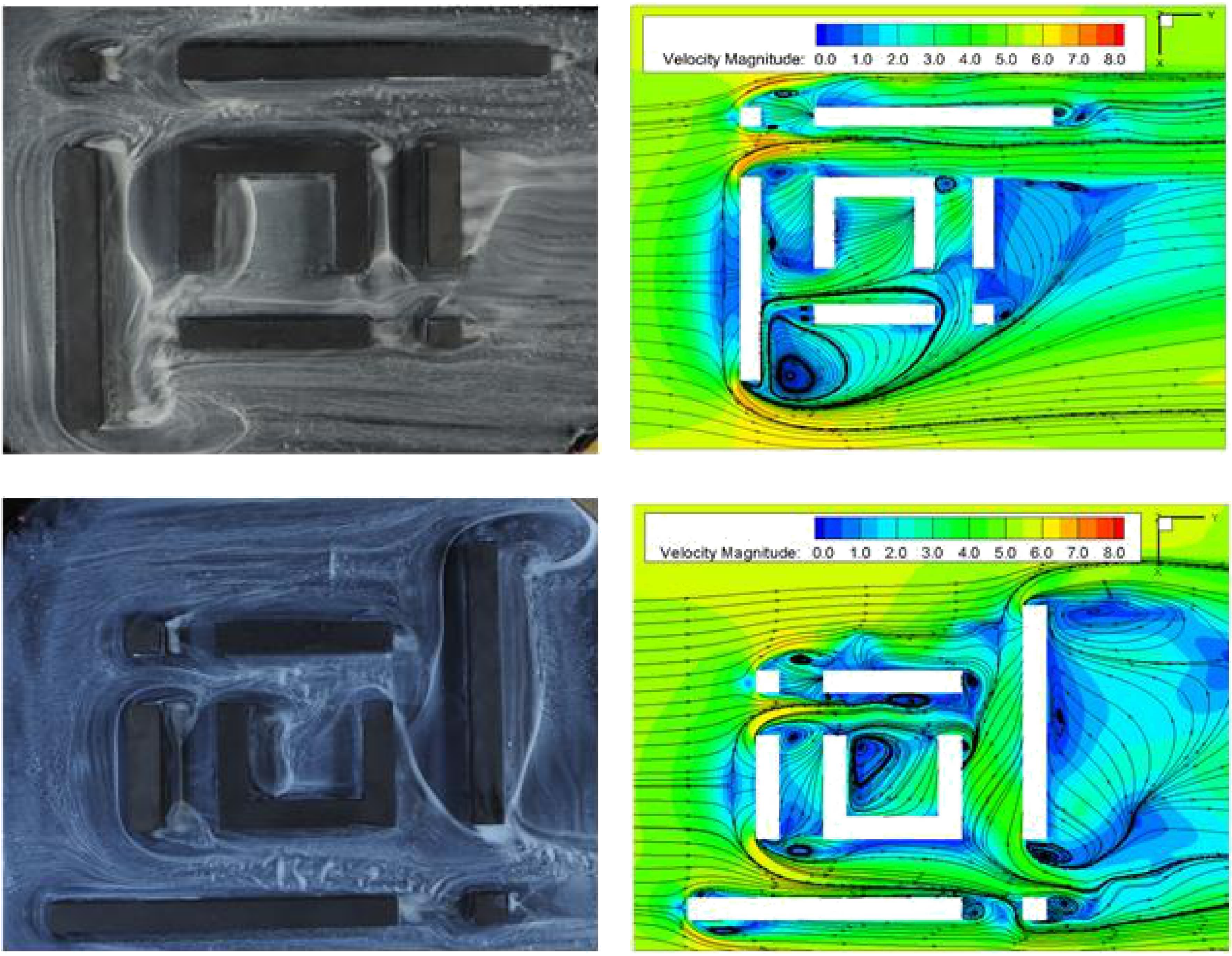

Fig. 6. Qualitative results of research for two different directions of flow: on the top – case 1, below – case 2; pictures on the left – oil flow visualization, on the right – numerical simulation showing velocity field in the plane $z = \frac{1}{18}h_b = 2.5$ mm (1 m in real dimensions)

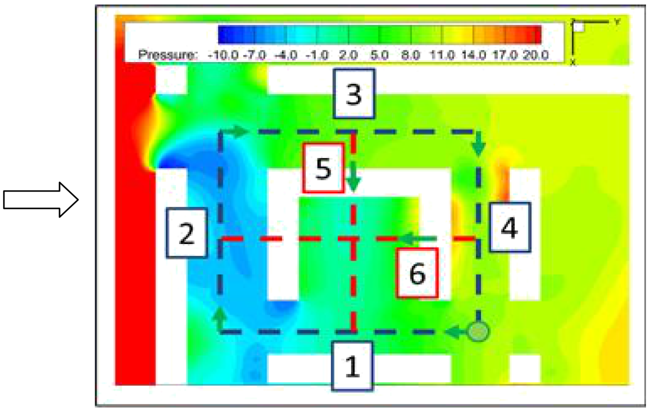

Fig. 7. Map of the pressure and paths of pressure measurement points in the plane $z = \frac{1}{18}h_b = 2.5$ mm, case 1 (direction of flow marked by the arrow on the left)

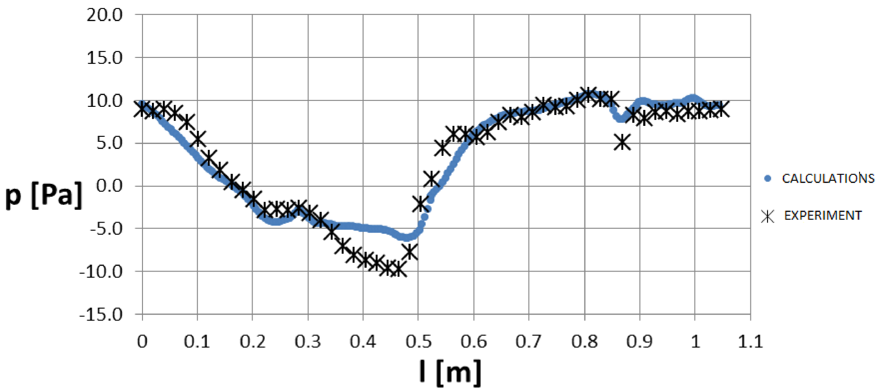

Fig. 8. Numerical and experimental results, path 1–4, case 1

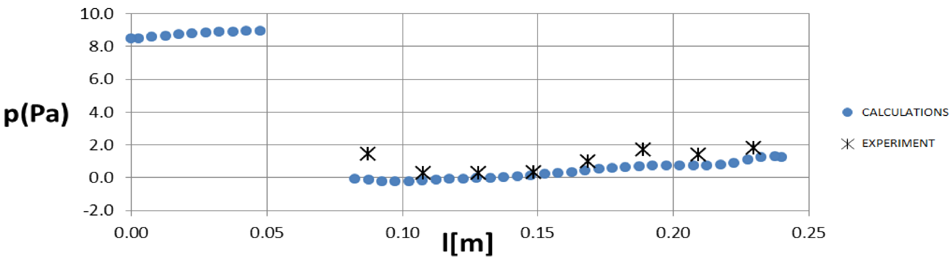

Fig. 9. Numerical and experimental results, path 5, case 1

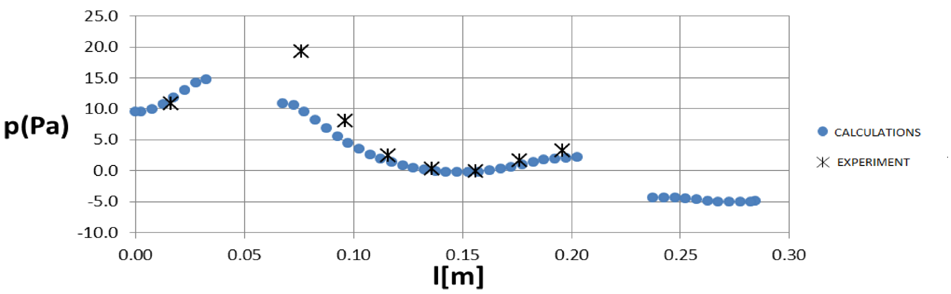

Fig. 10. Numerical and experimental results, path 6, case 1

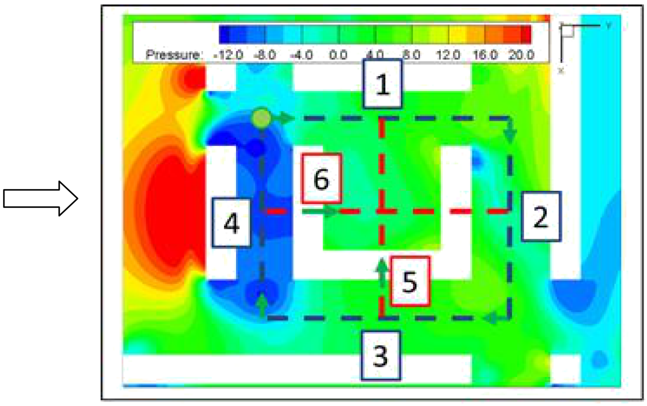

Fig. 11. Map of the pressure and paths of pressure measurement points in the plane $z = \frac{1}{18}h_b = 2.5$ mm, case 2 (direction of flow marked by the arrow on the left)

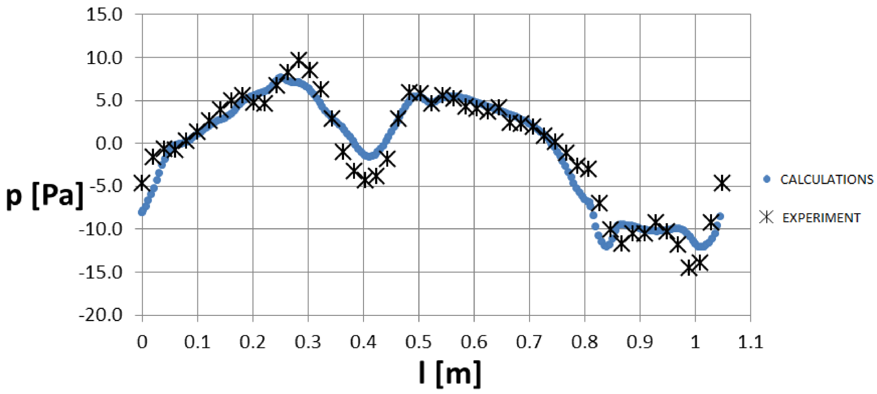

Fig. 12. Numerical and experimental results, path 1–4, case 2

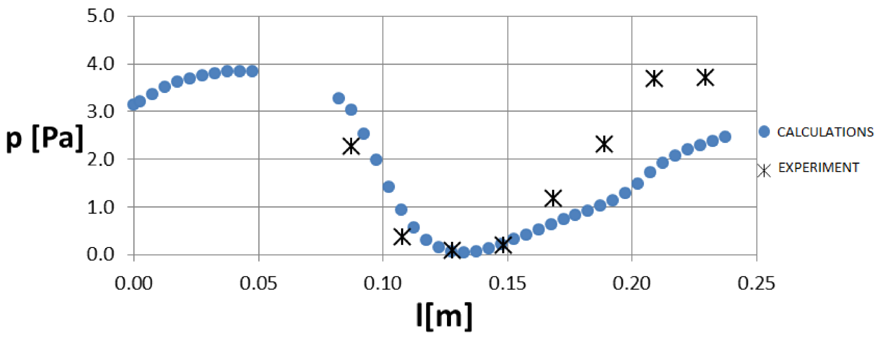

Fig. 13. Numerical and experimental results, path 5, case 2

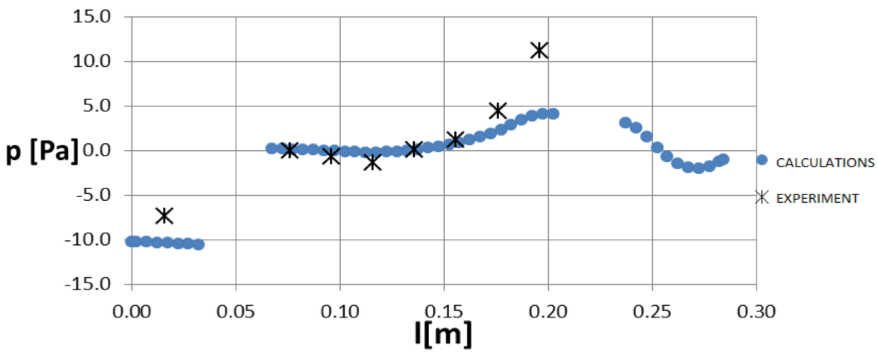

Fig. 14. Numerical and experimental results, path 6, case 2

In order to facilitate presentation of the quantitative results, the line segments along which the pressure measurements were made are numbered. Results of the analysis are presented in three separate graphs depicting pressures along segments 1-4 (rectangular loop), 5 and 6 (red sections). The independent variable is the length – measured along respective segments in the direction shown in the drawings (Fig.7).

The comparative analysis of the results showed that the selected turbulence model k-ε *realizable* allows for a relatively accurate determination of the pressure along the analyzed paths. The biggest differences are observed in the vicinity of the strongly turbulent zones. Pressure values obtained in other areas are almost identical with the results of the experiment conducted. The reason for discrepancies in the vicinity of zones marked out by strong turbulence is overproduction of the turbulence kinetic energy. It causes suppression of large vortices, which leads to reduction of pressure gradients. Difficulties associated with the correct determination of pressure in zones of strong turbulence and excessive turbulence intensity (in the area behind the buildings) are caused by the simplifications used in the model k-ε. Similar errors in the pressure distribution were obtained in other works [22]. It is worth mentioning that the pressure measurements conducted in a wind tunnel can also be affected by errors.

Both qualitative and quantitative results allow to conclude that the use of turbulence model k-ε *realizable* as well as the methodology for determining presuppositions for numerical simulation (velocity profiles and computational grid) presented in the paper give a reliable accurate representation of an airflow around complex geometric structures, similar to real buildings situations. Therefore they may be useful for town-planners and architects as a tool for optimization of the airflow around buildings and urban structures, especially in urban design and concept phase of architectural design, when considering natural or hybrid ventilation or mitigating the impact of Urban Heat Island.

6. Summary

This paper describes the results obtained from an analysis of an airflow around a sample of an urban development. This analysis contained:

- an investigation for several ways of discretization of calculation area and the choice of the best option,
- an attempt to find the closest match to the boundary conditions,
- comparison of calculation and experiment.

Before proceeding to the appropriate calculations a research was conducted to obtain the horizontal homogeneity of given numerically velocity profile. This was accomplished as suggested by the authors of the article [15].

Images of the flow patterns obtained by calculations are similar to oil visualization made for a given geometry. All distributions of pressure are close to the experimental measurements. The biggest differences in the results were observed in zones of strong turbulence. It is a typical problem that arises when the model of k-ε type is used to calculations [8, 23].

Despite some differences, the analysis performed in this work showed that numerical results obtained with the use of the k-ε *realizable* model of turbulence for quite complex geometry reflect the reality with sufficient for design practice accuracy. Air velocity and pressure maps obtained with the aid of the described methods are possible to be used as a design tool by architects and contain essential information (qualitative and quantitative) concerning aerodynamic phenomena around complex configuration of buildings. So, it can be assumed that modeling and simulation methodology presented in this work can be useful in the practice of urban and architectural design.

Numerical methods have not yet been developed sufficiently to completely replace the wind tunnel experiments. However, they may be sufficient for preliminary analysis of airflow around a building, when high accuracy is not required and can complement a more accurate tunnel research, greatly minimizing cost and time-consumption.

REFERENCES

[1] A. Flaga, *Wind Engineering*, Arkady, Warsaw, 2008, (in Polish).

[2] B. Blocken, "50 years of computational wind engineering. Past, present and future", *J. Wind Engineering and Industrial Aerodynamics* 129, 69–102 (2014).

[3] B. Blocken and J. Carmeliet, "Pedestrian wind environment around building: literature review and practical examples", *J. Thermal Envelope and Building Science* 28 (2), 107–159 (2004).

[4] K. Klemm, *A complex Assessment of Microclimate Conditions Found in Widely Spaced and Dense Urban Structures*, KILiW PAN, Warsaw, 2011, (in Polish).

[5] R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto, T. Nozu, and T. Shirasawa, "Cooperative project for CFD prediction of pedestrian wind environment in the architectural institute of Japan", *J. Wind Engineering and Industrial Aerodynamics* 95 (9–11), 1551–1578 (2007).

[6] D. Schulz, "Improved grid integration of wind energy systems", *Bull. Pol. Ac.: Tech.* 57 (7), 311–315 (2009).

[7] A. Mochida and I.Y.F. Lun, "Prediction of wind environment and thermal comfort at pedestrian level in urban area", *J. Wind Engineering and Industrial Aerodynamics* 96 (10–11), 1498–1527 (2008).

[8] K. Daniels, *The Technology of Ecological Building*, Birkhauser, Berlin, 1997.

[9] B. Blocken, J. Carmeliet, and T. Stathopoulos, "Application of CFD in building performance simulation for the outdoor environment: an overview", *J. Building Performance Simulation* 4 (2), 157–184 (2011).

[10] S.E. Kim and F. Boysan, "Application of CFD to environmental flows", *J. Wind Engineering and Industrial Aerodynamics* 81, 145–158 (1999).

[11] J. Franke, A. Hellsten, H. Schlünzen, and B. Carissimo, *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment*, University of Hamburg, Hamburg, 2007.

[12] J. Franke, A. Hellsten, H. Schlünzen, and B. Carissimo, "The COST 732 best practice guideline for CFD simulation of flows in the urban environment – A summary", *Int. J. Environmental Pollution* 44 (1–4), 419–427 (2011).

[13] S. Murakami, A. Mochida, and Y. Hayashi, "Examining the k-ε model by means of a wind tunnel test and large-eddy simulation of the turbulence structure around a cube", *J. Wind Engineering and Industrial Aerodynamics* 35, 87–100 (1990).

[14] D.A. Köse and E. Dick, "Prediction of the pressure distribution on a cubical building with implicit LES", *J. Wind Engineering and Industrial Aerodynamics* 98, 628–649 (2010).

[15] P.J. Richards and S.E. Norris, "Appropriate boundary conditions for computational wind engineering models revisited", *J. Wind Engineering and Industrial Aerodynamics* 99 (4), 257–266 (2011).

[16] D.A. Köse, D. Fauconnier, and E. Dick, "ILES of flow over low-rise buildings: Influence of inflow conditions on the quality of the mean pressure distribution prediction", *J. Wind Engineering and Industrial Aerodynamics* 99 (10), 1056–1068 (2011).

[17] S. Reiter, "Validation process for CFD simulations of wind around buildings", *Eur. Built Environment CAE Conf.* 1, CD-ROM (2008).

[18] A. Kovar-Panskus, P. Louka, J.F. Sini, E. Savory, M. Czech. A. Abdelqari, P.G. Mestayer, and N. Toy, "Influence of geometry on the mean flow within urban street canyons – a comparison of wind tunnel experiments and numerical simulation", *Water, Air and Soil Pollution* 2 (5–6), 365–380 (2002).

[19] B. Blocken and J. Persoon, "Pedestrian wind comfort around a large football stadium in an urban environment: CFD simulation, validation and application of the new Dutch wind nuisance standard", *J. Wind Engineering and Industrial Aerodynamics* 97 (5–6), 255–270 (2009).

[20] M. Sakr Fadl and J. Karadelis, "CFD Simulations for wind comfort and safety in urban area: a case study of coventry university central campus", *Int. J. Architecture, Engineering and Construction* 2 (2), 131–143 (2013).

[21] B. Blocken, J. Carmeliet, and T. Stathopoulos, "CFD simulation of the atmospheric boundary layer: wall function problems", *Atmospheric Environment* 41 (2), 238–252 (2007).

[22] E. Błazik-Borowa, *Difficulties Arising from the Use of K-Epsilon Turbulence Model for the Purpose of Determining the Airflow Around Buildings*, Lublin University of Technology Publisher, Lublin, 2008, (in Polish).

[23] ANSYS, Inc. *Ansys Fluent Theory Guide, version 14.0.*, ANSYS, Canonsburg, 2011.

[24] ANSYS Inc. *Ansys Fluent User's Guide, version 14.0.*, ANSYS, Canonsburg, 2011.

[25] R. Jóźwiak, "An analysis of a potential influence on airing and wind conditions of the area surrounding an urban layout planned to be built at a lot situated in Warsaw, Powązkowska street 23/1", in *Materials of Institute of Aeronautics and Applied Mechanics*, Warsaw University of Technology, Warsaw, 2013, (in Polish).