Computer Assisted Mechanics and Engineering Sciences, **16**: 309–319, 2009.

The quality of numerical modelling of the flow around two buildings with different heights

Ewa Błazik-Borowa
Department of Structural Mechanics, Lublin University of Technology
ul. Nadbystrzycka 40, 20-618 Lublin, Poland
e-mail: e.blazik@pollub.pl

(Received in the final form February 9, 2010)

The paper is devoted to the problems with quality of numerical modeling for two-dimensional incompressible flow around two models of buildings with different heights. The calculations have been made with use of the turbulence model $k-\varepsilon$ in the standard version and with the Finite Volume Method. The quality evaluation for the calculation is based on the comparison of the results with measurements in a wind tunnel. Hence, in this paper there have been presented the graphs of averaged velocities which are results of author's own measurements, as well as the graphs presenting the error in the calculated flow velocities. The main conclusion drawn from the research is that the flow around two models is more complicated than the flow around the single one and therefore the calculation results for the set of models are less accurate in comparison with the ones obtained for the single model.

1. Introduction

The paper is devoted to the problems with quality of numerical modelling for two-dimensional incompressible flow around two models with different heights. Such arrangements of two objects may be used to determine wind action on the walls of buildings in an urban space. The neighborhood of the surrounding building is taken into consideration during the calculation of the value of wind load on the standards basis by introduction of the roughness of ground and the shape of the velocity profile. Whereas the standard recommendations do not include the local flow around particular buildings, for example, narrow spaces between buildings or the air suction at flow above an urban canyon. The standard approximation is enough for heavy objects, but when designing building scaffoldings or light building elevations the wind action ought to be more exactly calculated. It may be made by using computational analysis or measurements in wind tunnels. The calculations with use of the methods of computer fluid dynamic seem to be the better solution of the problem because the measurements are more expensive and they need much more time.

The methods of CFD should be verified by research in wind tunnels. It is particularly important in the case of the flow parameters calculations because these methods introduce more simplifications than methods of the solid mechanics. Moreover the equations which describe parameters of a flow, for example turbulence models, usually contain semi-empirical coefficients. The number of these coefficients and their values depend on the type of flow (for example, compressible or incompressible flow) and turbulence model problems. It means that the flow exercises should be checked at every steps of research. The verification may be made, for example, on the basis of the comparison of the calculation and measurements results or with use of the solution sensitivity analysis to the model parameters. The problems of the quality of numerical modelling in civil engineering with use of the sensitivity analysis are presented in the papers [1] and [2]. Here the evaluation of the calculation quality is made on the basis of the comparison with measurements in a wind tunnel. The calculations have been performed by using the turbulence model $k-\varepsilon$ in standard version and the Finite Volume Method (FVM). The results of computer calculations, presented in this research, have been obtained with use of FLUENT. The measurements have been carried out in the wind tunnel of the Wind Engineering Laboratory in Cracow University of Technology. The similar problems for buildings, but with the equal heights, are presented in many papers, for example, [5] and [10].

2. The description of building models

2.1. The set of models in the wind tunnel

The analyses have followed the measurements which had been carried out in the wind tunnel of the Wind Engineering Laboratory in Cracow University of Technology (Fig. 1).

Fig. 1. The arrangements of models in measurements space in the wind tunnel

The models had been set at the ground of the wind tunnel and the flow in the middle-plane could be treated as the two-dimensional one. The measurements had been provided for the set of two models: smaller cylinder of the square cross-section with dimensions $h \times h \times b$ and for the larger one with dimensions $2h \times h \times b$, where: $h = 200\,\text{mm}$ and $b = 2050\,\text{mm}$. The models arrangements have been presented in Fig. 1 and Table 1.

Table 1. The sets of models

Set	Distance L [m]	Location of the square cylinder
b		single square
c		single rectangular
d	0.4	in front of the rectangular
e	0.6	in front of the rectangular
f	0.8	in front of the rectangular
g	0.4	behind the rectangular
h	0.6	behind the rectangular
i	0.8	behind the rectangular

2.2. The numerical model

The calculations have been made for the two-dimensional and incompressible problem with use of the turbulence model $k-\varepsilon$ in the standard version. Besides the continuity and Navier-Stokes equations, this model contains two equations: the turbulence kinetic energy and the dissipation rate of the kinetic turbulence energy ones. The equations of the standard turbulence model $k-\varepsilon$ can be described by following formulae ([2], [9]):

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{1}$$

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + 2\frac{\partial}{\partial x_m}\left[\left(\mu+\mu_t\right)s_{im}\right], \tag{2}$$

$$\rho\left(\frac{\partial k}{\partial t} + u_j\frac{\partial k}{\partial x_j}\right) = \frac{\partial}{\partial x_m}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_m}\right] + \rho P_k - \rho\varepsilon, \tag{3}$$

$$\rho\left(\frac{\partial \varepsilon}{\partial t} + u_j\frac{\partial \varepsilon}{\partial x_j}\right) = \frac{\partial}{\partial x_m}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_m}\right] + C_{\varepsilon 1}\rho P_k\frac{\varepsilon}{k} - \rho C_{\varepsilon 2}\frac{\varepsilon^2}{k}, \tag{4}$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, \tag{5}$$

where: p – pressure, u_i – velocity components, k – kinetic turbulence energy, e – dissipation of a kinetic turbulence energy, μ_t – turbulence dynamic viscosity, t – time, ρ – density, μ – dynamic viscosity, ρP_i – turbulence energy production, $s_{im} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_m} + \frac{\partial u_m}{\partial x_i}\right)$ – strain rate.

The above relationships were described in the papers [8] and [9]. This model is, at present, the most popular model among the users of commercial CFD programs. This is not the effect of its perfection, because the model has many faults, but just because the shortcomings of the model are well known and tested. One of the main and unfortunately probably still not solved problems of the $k-\varepsilon$ model is the selection of semi-empirical coefficients. Getting the correct solution by the turbulent model $k-\varepsilon$ in standard version depends, among others, on the selection of five “constants”: C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε. These values are the result of non dimensional theoretical analyses and much research, among others in the wind tunnels, and that is why they are called “semi-empirical coefficients” or “constants”. The process of choosing their values is called the calibration of the model and it is described in such works as: [2], [7, 8] and [13]. In literature, they are called “constants” of model $k-\varepsilon$, but it seems to be the incorrect name, because there can be found various values of these parameters when studying the literature (e.g. [6–9, 12, 14]).

In this paper two sets of semi-empirical coefficients of the $k-\varepsilon$ model are used and they are:

1. $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$ – the set of “constants” most often used and called the standard set,
2. $C_{\varepsilon 1} = 0.36$, $C_{\varepsilon 2} = 0.82$, $C_\mu = 0.042$, $\sigma_k = 0.24$, $\sigma_\varepsilon = 1.03$ – the set obtained on the basis of the calibration process which is described in the paper [2] and called here the new set.

The proper assumption of boundary conditions is very important in calculations. Here, the following boundary conditions are assumed:

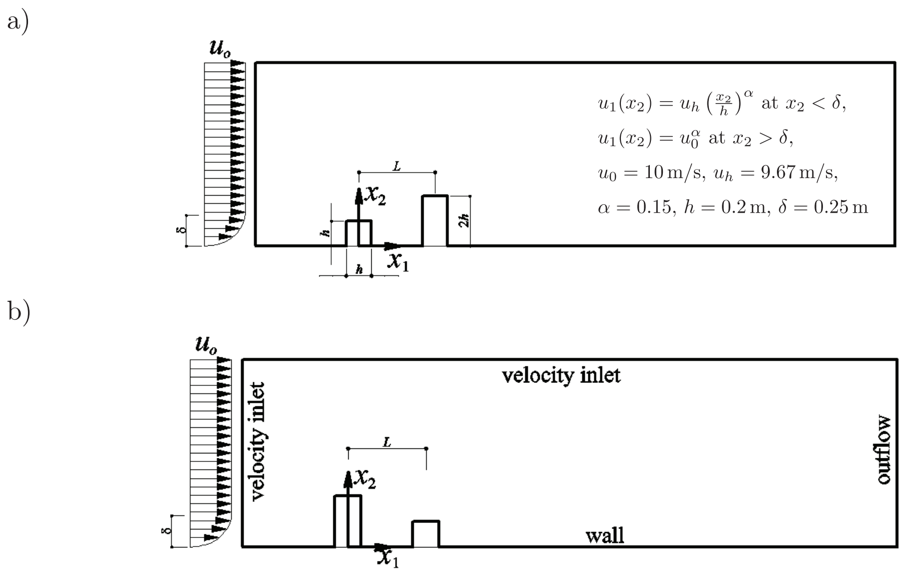

Fig. 2. The arrangements of models: a) the set with a square as the upstream model and the description of inlet velocity profile, b) the set with a square as the downstream model and boundary conditions

1. at the upstream edge the inflow velocity is described by a power function (Fig. 2), turbulence kinetic energy and its dissipation are calculated from turbulence intensity on the basis of the results of measurements in the wind tunnel,
2. the bottom edge is modeled as the wall with no-slip conditions,
3. the upper edge is modeled as inflow boundary with the constant values of flow parameters,
4. and the downstream edge is assumed as the outflow boundary.

The calculation domain of the problem with boundary conditions is shown in Fig. 2.

3. The analysis of results

3.1. The measurements results

The variations in time of pressure distributions on model walls and time series of velocities in eight cross-sections of the wind tunnel measuring space have been obtained as the result of the measurements. The measurements of pressures on models have been obtained with use of pressure scanner based on piezoresistive two-directional pressure sensors Motorola MPX 2010 and battery of manometers. The measurements carried with use of pressure scanner have been made in 30 points distributed over the model surface and with the sampling frequency of 500 Hz. Additional measurements have been provided with use of hot-wire anemometers with the sampling. The time series of velocity components along directions perpendicular to the wires have been obtained on the basis of measurements. The final components of velocity vector along mean flow and in the perpendicular direction have been calculated according to the recommendations which can be found in the papers such as [2], [10] and [12]. Such procedure allows removal of the influence of the turbulence in the direction perpendicular to the measurement plane and the influence of wire cooling due to flow along the wire direction.

The errors analysis is presented in the paper [2]. Here, the measurements results for pressure are shown in Figs. 3 and 4, the results for velocity components are shown in Figs. 5 to 8.

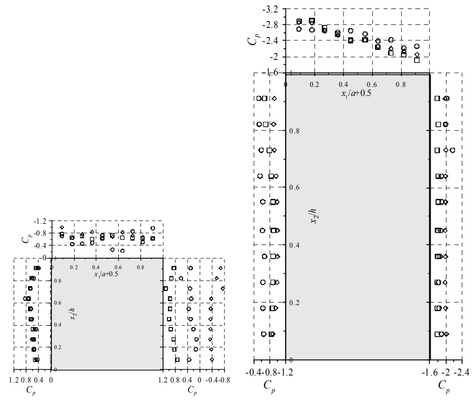

Fig. 3. The aerodynamic coefficient C_p (where $C_p = \frac{p}{0.5\rho u_0^2}$) for measurements with the rectangular model as the upstream one: ○ – set d, □ – set e, ◇ – set f

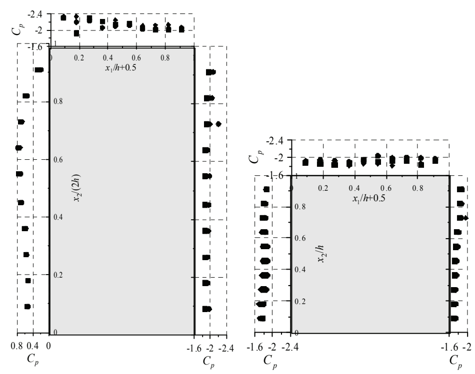

Fig. 4. The aerodynamic coefficient C_p (where $C_p = \frac{p}{0.5\rho u_0^2}$) for measurements with the rectangular model as the upstream one: ● – set g, ■ – set h, ◆ – set i

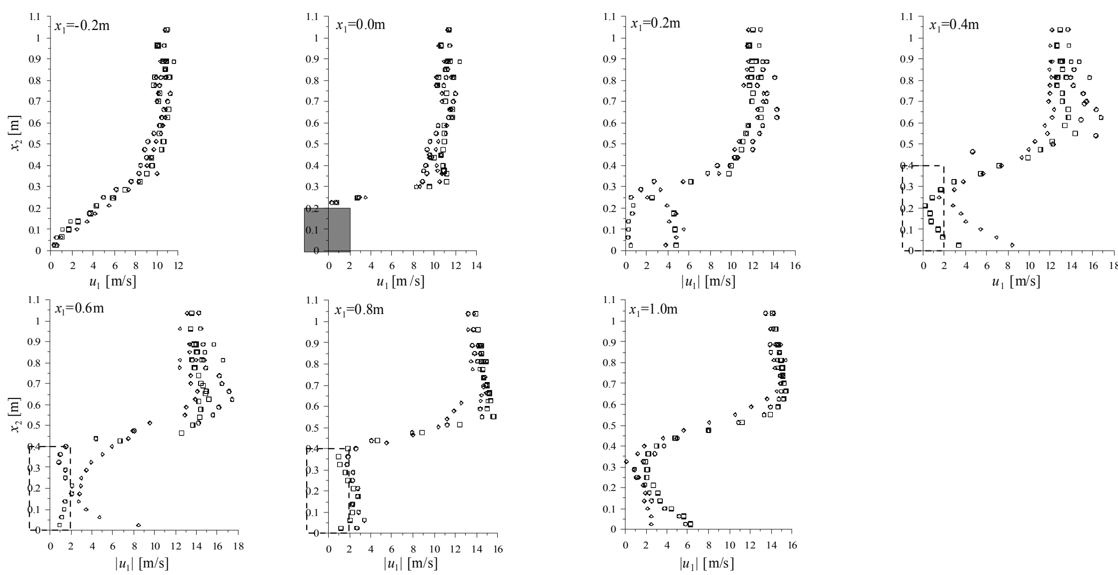

Fig. 5. The components $|u_1|$ of velocity vectors along the averaged direction of an inflow obtained by measurements: ○ – set d, □ – set e, ◇ – set f

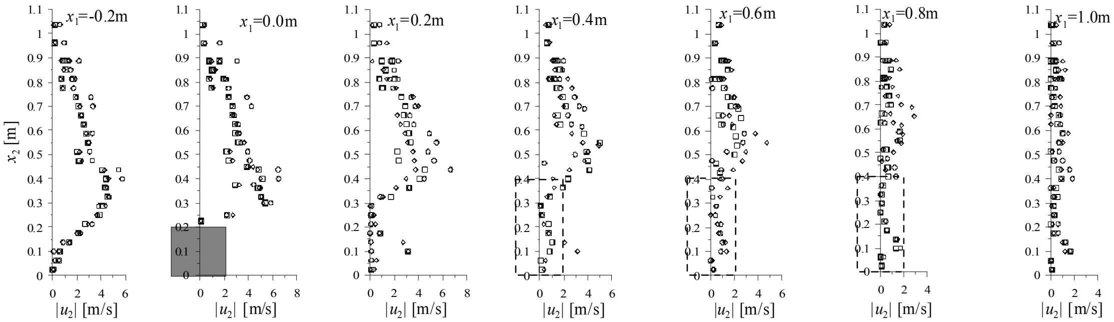

Fig. 6. The components $|u_2|$ of velocity vectors perpendicular to the averaged direction of an inflow obtained by measurements: ○ – set d, □ – set e, ◇ – set f

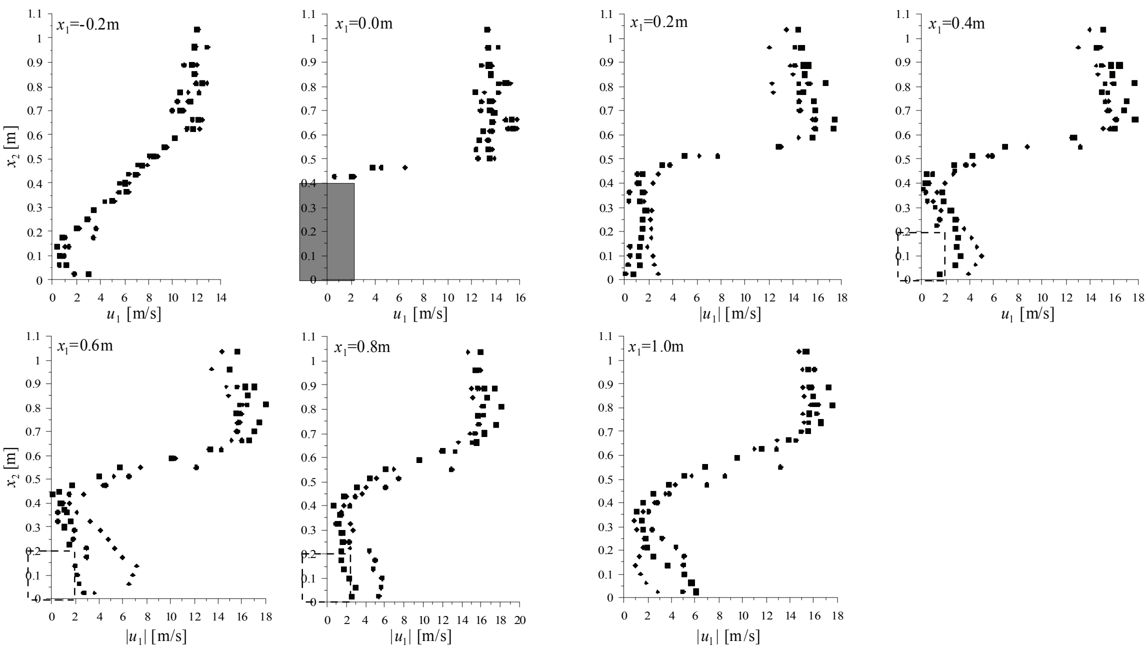

Fig. 7. The components $|u_1|$ of velocity vectors along the averaged direction of an inflow obtained by measurements: ● – set g, ■ – set h, ⧫ – set i

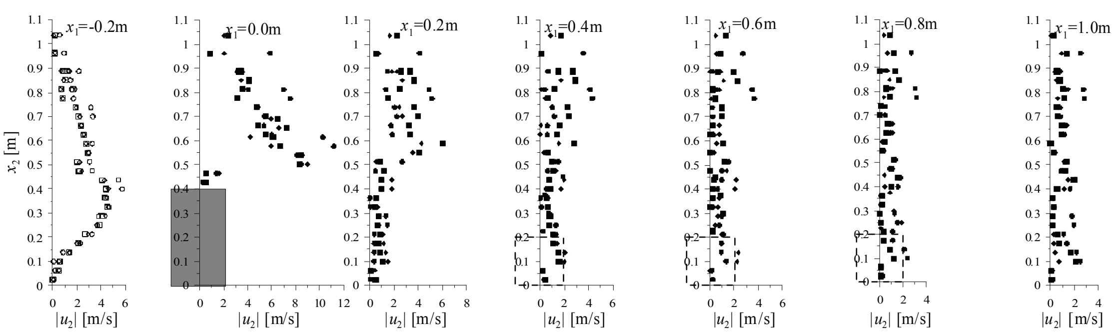

Fig. 8. The components $|u_2|$ of velocity vectors perpendicular to the averaged direction of an inflow obtained by measurements: ● – set g, ■ – set h, ⧫ – set i

3.2. The calculation results

The calculation results contain averaged values of pressure, velocities, turbulence kinetic energy, its dissipation and dynamic turbulence viscosity. The fields of velocity components for two sets of the turbulence model coefficients are shown in Figs. 9 and 10.

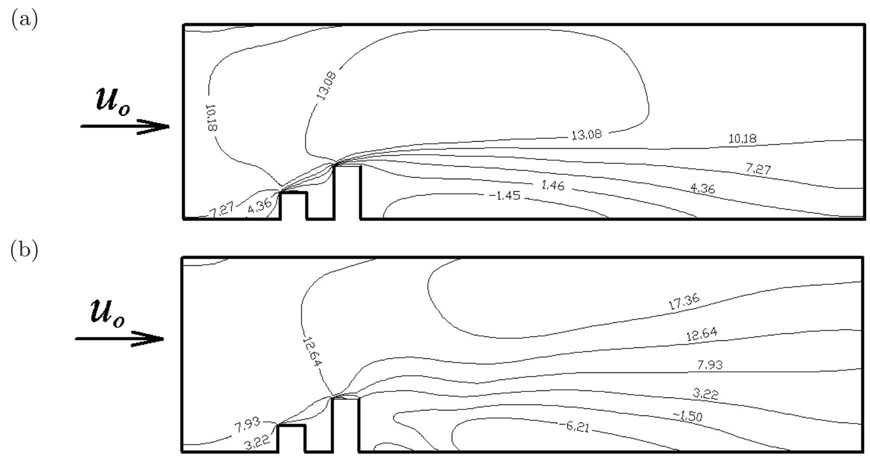

Fig. 9. The components u_1 [m/s] of velocity vectors along the averaged direction of an inflow obtained by calculations: (a) at the coefficients: $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, (b) at new set of the coefficients: $C_{\varepsilon 1} = 0.36$, $C_{\varepsilon 2} = 0.89$, $C_\mu = 0.042$, $\sigma_k = 0.24$, $\sigma_\varepsilon = 1.03$

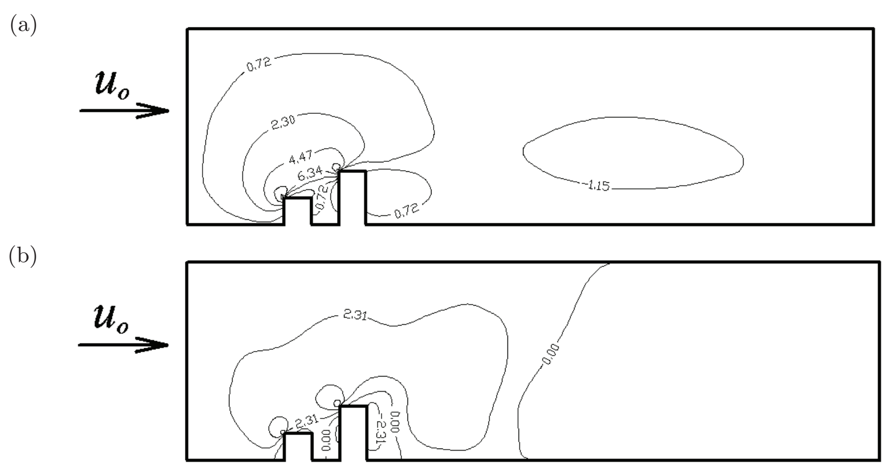

Fig. 10. The components u_2 [m/s] of velocity vectors perpendicular to the averaged direction of an inflow obtained by calculations: (a) at the coefficients: $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, (b) at new set of the coefficients: $C_{\varepsilon 1} = 0.36$, $C_{\varepsilon 2} = 0.89$, $C_\mu = 0.042$, $\sigma_k = 0.24$, $\sigma_\varepsilon = 1.03$

3.3. The comparison of measurements and numerical results

The results of measurements and calculations are the pressure distributions at walls of the models and the velocity components in the points of the research space. Here, the comparison of the values of the velocity are performed at points only. The pressure distributions are not analyzed, because these results can be incorrect. The proper determination of the pressure distribution is possible when the pressure values are obtained from calculations with use of the pressure boundary conditions (cf. Eqs. (1) to (5)). For incompressible flow the boundary conditions of the pressure are not used, which results in the fact that only the pressure gradients are correctly calculated and the pressure values are often not properly evaluated. It does not influence the other results of the calculation, but the errors of pressure depend strongly on the description of the velocity field. This problem is described in papers [4] and [11].

In the paper [2] the juxtapositions of velocities obtained from measurements and calculations at two sets of model coefficients are shown. Here, the errors of resultant velocity values in the cross-section above models are shown in Figs. 11 and 12 and the errors in cross-section at the $2h$ distance from the middles of models are shown in Figs. 13 and 14.

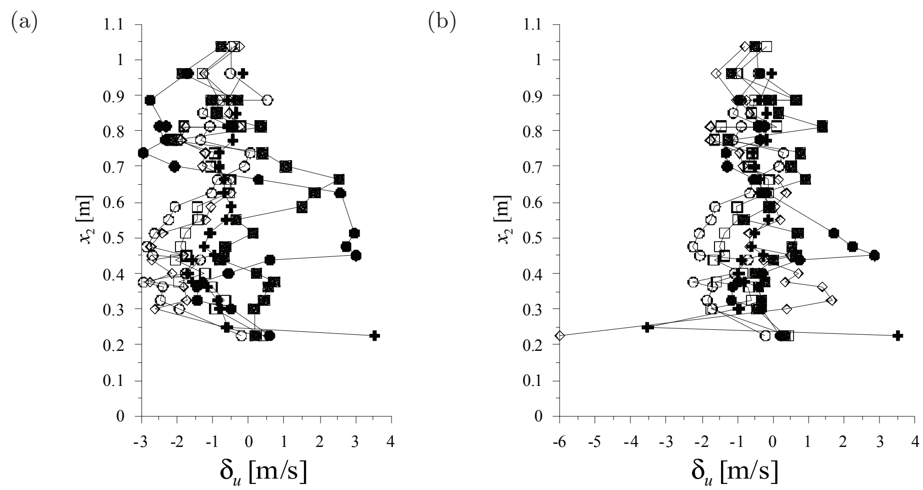

Fig. 11. The calculation errors of velocities in the vertical cross-section above the square model for: (a) the standard set of "constants", (b) a new set of "constants": ✚ – set b ($x_1/h = 0$), ○ – set d ($x_1/h = 0$), □ – set e ($x_1/h = 0$), ◇ – set f ($x_1/h = 0$), ● – set g ($x_1/h = 2$), ■ – set h ($x_1/h = 3$)

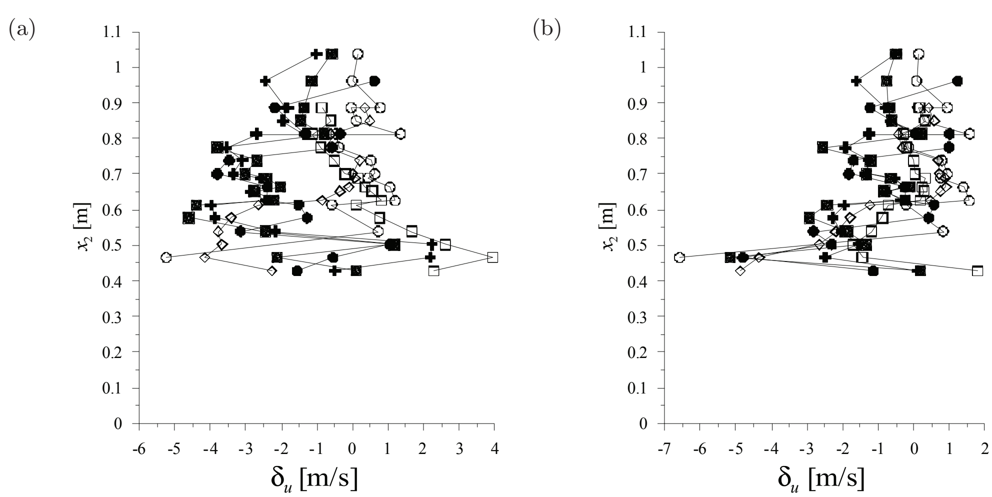

Fig. 12. The calculation errors of velocities in the vertical cross-section above the rectangular model for: (a) the standard set of "constants", (b) a new set of "constants": ✚ – set c ($x_1/h = 0$), ○ – set d ($x_1/h = 2$), □ – set e ($x_1/h = 3$), ◇ – set f ($x_1/h = 4$), ● – set g ($x_1/h = 0$), ■ – set h ($x_1/h = 0$)

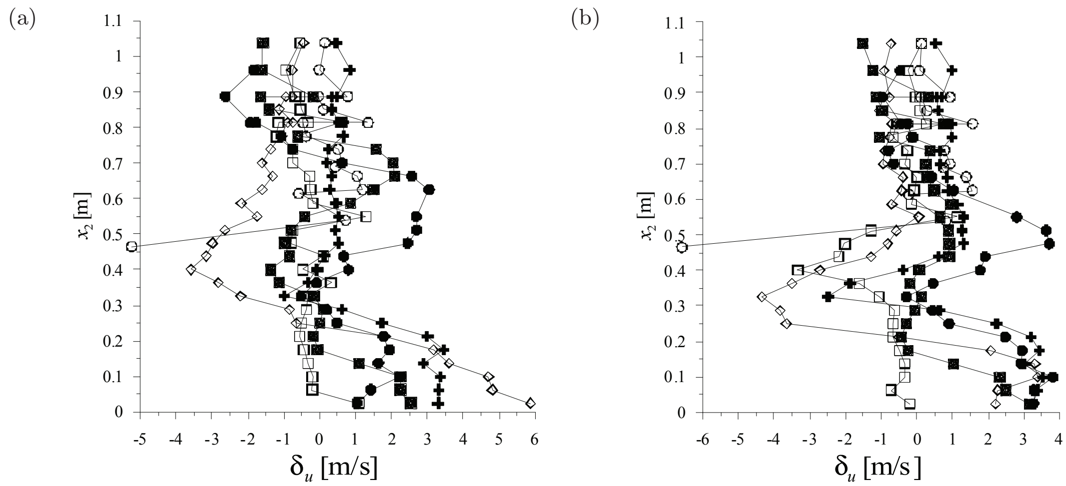

Fig. 13. The calculation errors of velocities in the vertical cross-section at the $2h$ distance from the middle of square for: (a) the standard set of "constants", (b) a new set of "constants": ✚ – set b ($x_1/h = 2$), ∘ – set d ($x_1/h = 2$), □ – set e ($x_1/h = 2$), ⋄ – set f ($x_1/h = 2$), • – set g ($x_1/h = 4$), ■ – set h ($x_1/h = 5$)

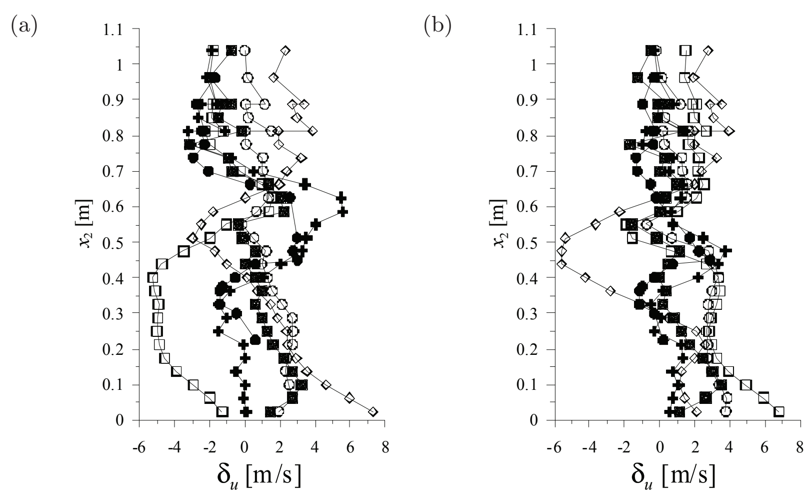

Fig. 14. The calculation errors of velocities in the vertical cross-section at the $2h$ distance from the middle of rectangular for: (a) the standard set of "constants", (b) a new set of "constants": ✚ – set c ($x_1/h = 2$), ∘ – set d ($x_1/h = 4$), □ – set e ($x_1/h = 5$), ⋄ – set f ($x_1/h = 6$), • – set g ($x_1/h = 2$), ■ – set h ($x_1/h = 2$)

These errors are calculated as differences between measurements and calculations results at points of the research space. On the basis of these comparisons it can be noted that:

- errors of the velocity values behind models are bigger than errors above models,
- errors behind the upstream object are smaller than errors behind the same model located as downstream one,
- results for velocity fields improve when the models are well separated, it comes from the fact that the common influence between models and flow is smaller for bigger distance between objects,
- results may be improved by using the $k - \varepsilon$ model coefficients which are calculated from the process calibration, in the analysed problem the errors are about 20 % smaller for the new set of coefficients.

The height of the upstream building has significant influence on quality of results. It is seen that for the set of two models the errors for cases with the rectangular as upstream object are bigger than for exercises with squares as upstream ones. The values of velocities in cross-section in front of the upstream model are too big. At the upper edge of the smaller model the difference between calculations and measurements equals about 2 m/s (while mean inflow velocity is equal to 10 m/s) and for bigger model this value is bigger and it is equal to 4 m/s. These errors do not come from the $k-\varepsilon$ model and they are caused by using the numerical method called upwind scheme (cf. [11]). In FVM the flow parameters are calculated in the middle of cells. The values at edges are also needed in calculations. In the upwind scheme they can be assumed as equal to value in the upwind cell or approximated by linear functions on the basis value in the upwind cells. These errors do not depend on the model coefficients and they cannot be decreased by calibration. It is also seen in the sensitivity analysis of flow parameters to the model coefficients which is presented in the paper [2]. The sensitivity coefficients in the space in front of the model are closed to zero.

4. Conclusion

The main conclusion coming from the research is that the flow around two models is more complex than the flow around the single one. It is caused by two reasons:

- errors of the $k-\varepsilon$ model, which arise at an upstream model, are added to errors at a downstream model,
- there are Reynolds stresses between models which are bigger than values which are acceptable in the $k-\varepsilon$ model.

Additionally, significant errors occur in front of the upstream model. This comes from the use of the upwind scheme. Such method is used together with the FVM and it depends on the assumption that the values of flow properties at edges are equal to the values from upstream adjacent volumes. It also causes the incorrect calculation results in front of the high buildings.

The errors which come directly from the $k-\varepsilon$ model may be decreased by changing the model "constants". Unfortunately, the modified values of "constants" can cause the instability of iteration process during the calculations and therefore this problem will be developed in future research.

References

[1] E. Błazik-Borowa. The sensitivity analysis of the flow around a square cylinder to parameters of the $k-\varepsilon$ method. In: *Proceedings of the 12th International Conference on Wind Engineering*, Cairnes, Australia, 2007, Vol. II, pp. 1919–1926.

[2] E. Błazik-Borowa. *The problems of the application of the $k-\varepsilon$ turbulence model to determination of the properties of flow around buildings*. The Publisher of Lublin University of Technology, 2008.

[3] E. Błazik-Borowa. Analysis of the channel flow sensitivity to the parameters of the $k-\varepsilon$ method. *International Journal for Numerical Methods in Fluids*, **58**: 1257–1286, 2008.

[4] E. Błazik-Borowa, J. Podgórski. The choice of boundary conditions of the $k-\varepsilon$ method for pressure (in Polish). In: *Proceedings of the XII Polish Conference "Building physics in theory and practice"*, 2009, pp. 15–16.

[5] S.R. Hanna, S. Tehranian, B. Carissimo, R.W. Macdonald, R. Lohner. Comparisons of model simulations with observations of mean flow and turbulence within simple obstacle arrays *Atm. Env.*, **36**: 5067–5079, 2002.

[6] C.M. Hrenya, E. J. Bolio, D. Chakrabarti, J. L.Sinclair. Comparison of low Reynolds number $k-\varepsilon$ turbulence models in predicting fully developed pipe flow. *Chemical Engineering Science*, **50**(12): 1923–1941, 1995.

[7] A.V. Johansson. Engineering turbulence models and their development, with emphasis on explicit algebraic Reynolds stress models. In: M. Oberlack, F.H. Busse, eds. *Theories of Turbulence*. CISM Courses and Lectures, **442**, pp. 253–300. Springer Wien New York, 2002.

[8] B.E. Launder, D.B. Spalding. *Mathematical models of turbulence*. Academic Press, London and New York, 1972.

[9] B.E. Launder, D.B. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mech. and Eng.*, **3**: 269–289, 1974.

[10] F.S. Liena, E. Yeeb, Y. Chenga. Simulation of mean flow and turbulence over a 2D building array using high-resolution CFD and a distributed drag force approach. *J. of Wind Eng. and Ind. Aerodynamics*, **92**: 117–158, 2004.

[11] S.V. Patankar. *Numerical heat transfer and fluid flow*. McGraw-Hill Book Company, 1980.

[12] S.G. Sajjadi, M.N. Waywell. Application of roughness-dependent boundary conditions to turbulent oscillatory flows, *Int. Journal Heat and Fluid Flow*, **18**: 368–375, 1997.

[13] T.H. Shih, W.W. Liou, A. Shabbir, Z. Yang, J. Zhu. A new $k-\varepsilon$ eddy-viscosity model for high Reynolds number turbulent flows. *Computers Fluid*, **24**(3): 227–238, 1995.

[14] R.M.C. So, H.S. Zhang, C.G. Speziale. Near-wall modeling of the dissipation rate equation. *AIAA Journal*, **29**: 2069–2076, 1991.