ROCZNIKI INŻYNIERII BUDOWLANEJ – ZESZYT 15/2015
Komisja Inżynierii Budowlanej Oddział Polskiej Akademii Nauk w Katowicach

INFLUENCE OF NEARBY STRUCTURE ON THE WIND FLOW AROUND THE CUBE STRUCTURE

Ivana OLEKŠÁKOVÁ, Oľga HUBOVÁ, Lenka KONEČNÁ
Faculty of Civil Engineering, Slovak University of Technology in Bratislava, Slovakia

Keywords: *wind tunnel, turbulent wind flow, cube structure, pressure coefficient, experimental measurement.*

1. Introduction

The wind tunnel modeling of low-rise buildings differs from the modeling of high-rise buildings. A literature recommends using of large scale (at least 1:50) for modeling of low-rise buildings. At present, it is very popular to use CFD (Computational Fluid Dynamics) simulations because they are faster and cheaper than wind tunnel testing. The problem is that CFD is not suitable for all types of problems. The accuracy of the results depends on chosen mathematical model, quality of meshing, horizontal homogeneity of boundary layer, etc. Therefore, it is necessary to verify the results obtained by CFD simulations with the results from the tests, the literature and valid standards.

2. Turbulent flow over a bluff body

This paper deals with the influence of nearby structure on the wind flow around a rigid body. It belongs to the most important problems in the aerodynamics of buildings. Especially in the work, the turbulent wind flow over a 3D structure was investigated [1]. The character of this turbulent flow onto a 3D building can be clearly understood from Fig. 1a.

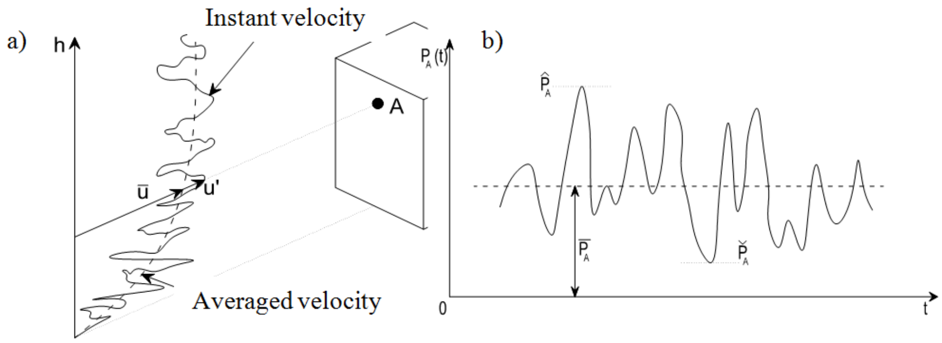

Fig. 1. a) Turbulent flow onto a 3D cube. b) Pressure fluctuations at point A.

Wind velocity profile is obtained after averaging of all measured data from long time period that is compared with the turbulent fluctuations. It lasts usually 20 minutes in full-scale or 1 minute in wind tunnel. It depends on valid codes and standards. This flow varies in time and in space. This fact causes fluctuating of the pressures at all points of the surface in time. Then the pressure time history at given point (point A, Fig. 1b) in time domain, is obtained. From this signal, several pressure values can be determined. Eq. 1 defines *the mean wind pressure* using the mean dynamic pressure as the reference value:

$$\overline{P}_A - P_\infty = \overline{C}_p \frac{1}{2} \rho \overline{U}^2 . \tag{1}$$

Eq. 2 defines *the peak pressure* by using the mean dynamic pressure and Eq. 3 defines *the peak pressure* by using the peak dynamic pressure:

$$\hat{P}_A - P_\infty = \hat{C}_p \frac{1}{2} \rho \overline{U}^2 , \tag{2}$$

$$\hat{P}_A - P_\infty = \hat{\hat{C}}_p \frac{1}{2} \rho \hat{U}^2 . \tag{3}$$

In the case of the measurements made in the wind tunnel, it is very important to determine the reference wind velocity. Pressure coefficients are very often related to the reference wind velocity at the top of the building. During the measurement the model is removed and measuring probe is placed in the height of the building's roof. External wind pressure coefficients are important for the determination of the wind effects on designed or advised structures. In this article, they were obtained in this way:

$$\overline{P}_A - P_\infty = \overline{C}_{ph} \frac{1}{2} \rho \overline{U}_h^2 . \tag{4}$$

3. Description of the experimental measurements

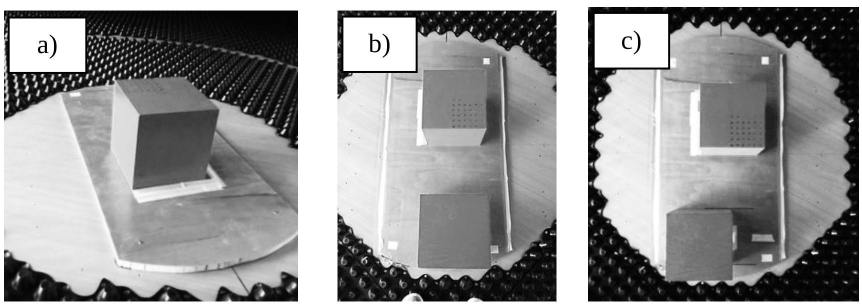

Fig. 2. Cube positions - a) Solitary cube - MC0, b) Position MC1, c) Position MC2.

The experimental measurements were made in Boundary Layer Wind Tunnel (BLWT) in Bratislava. Three different positions of cubes were taken into account (Fig. 2). At first, only self-standing cube with the dimensions of (200×200×200) mm, was tested (Fig. 2a). The model scale was 1:30. The results obtained on the model were compared with the values obtained from the measurements made in-situ on real SILSOE cube (the dimensions were 6×6×6 m) [2,4]. In our case, 32 pressure taps were placed on the roof of the cube. The measurements were repeated for all 3 positions: self-standing cube MC0 (Fig. 2a); examined cube exactly behind the shielding cube – position MC1 (Fig. 2b); shielding cube placed about 100 mm on the left side of the examined cube - position MC2 (Fig. 2c). The reference wind velocities were chosen with respect to flow similarity of the prototype and the model [1-3]. The measurements were repeated for 2 different wind velocities (7.5 [m/s] and 10.75 [m/s]) and for four angles of rotations – 0°, 15°, 30° and 45° (Fig. 3) in clockwise direction. The change of the wind flow on the roof of the cube was examined.

4. Evaluation of the results from the measurements

In the case of self-standing cube, the obtained results were compared with the literature. They confirmed the extreme value of the wind pressure coefficient on the roof of the structure at the angle of rotation 45º (C_P was -2.404). In comparison with the position MC1, the coefficients of suction on the roof determined for all wind directions did not reach these extreme values. In the case of the position MC2, the suction increased when the angles of rotation were 15º and 30º.

Table 1. Values of pressure coefficients for all angles of rotation and all positions.

Wind direction	MC0 max	MC0 min	MC1 max	MC1 min	MC2 max	MC2 min
0°	-0.584	-1.137	-0.111	-0.299	-0.199	-1.054
15°	-0.316	-1.603	-0.165	-1.224	-0.317	-1.708
30°	-0.291	-2.006	-0.261	-1.930	-0.370	-2.156
45°	-0.284	**-2.404**	-0.344	-2.104	-0.365	-2.177

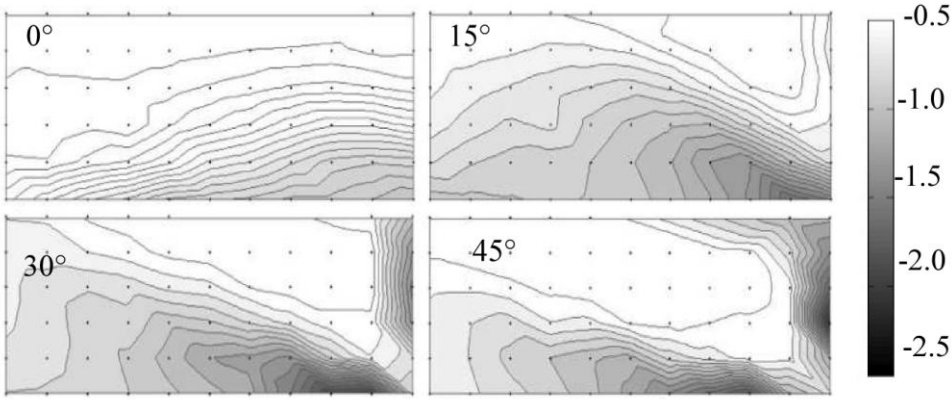

Fig. 3. Exemplary contour maps of C_p for wind directions of 0°, 15°, 30°, 45° for MC1.

Then, it can be said that there are no larger values of suction on the roof than it is in the case of alone-standing cube ($C_{p,1}$=-2.5, according to EN 1991-1-4), at angle of rotation 45°. Other positions MC1 and MC2 did not cause any larger values of suction. For the comparison, the maximum value of pressure coefficient determined according to Eurocode, for the cube and angle of rotation 45° is equalled to -2.5. It is in a good agreement with the results obtained from the measurements (C_P was -2.404, Table 1).

5. CFD simulation

Simulation software OpenFOAM and *k*-*ε* model were used for the calculation. It belongs to the category of RANS two-equation models. These classical models are based on Reynolds Averaged Navier-Stokes equations (time averaged). They are divided into the zero-equation models, one-equation models, two-equation models and seven equation models [5]. Many turbulence models are based on the Boussinesq hypothesis and it was proposed that the Reynolds stresses could be linked to the mean rate of deformation [4,5]. Using the suffix notation where *i, j* and *k* denote *x, y* and *z* directions respectively, viscous stresses are given by the following equation:

$$\tau_{ij} = \mu e_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{5}$$

and Reynolds stresses are linked to the mean rate of deformation:

$$\tau_{ij} = -\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right). \tag{6}$$

The most important part of creating of the model is the prediction of the turbulent viscosity, which gave very large number of possibilities for mathematical modeling. One of them is *k*-*ε* model which belongs to the two-equation models, where *k* and *ε* are turbulent kinetic energy and dissipation rate of *k* respectively. The instantaneous kinetic energy $k(t)$ of a turbulent flow is the sum of mean kinetic energy *K* (given by Eq. 7) and turbulent kinetic energy *k* (given by Eq. 8) and the relation for turbulent viscosity ν_t can be expressed by Eq. 10.

$$K = \tfrac{1}{2}\left(U^2 + V^2 + W^2\right), \tag{7}$$

$$k = \tfrac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right), \tag{8}$$

$$k(t) = K + k\,, \tag{9}$$

$$\nu_t \propto \frac{k^2}{\varepsilon}\,. \tag{10}$$

Then, the turbulent kinetic energy *k* is defined by the following equation:

$$\frac{\partial(\rho k)}{\partial t} + \mathrm{div}(\rho k\mathbf{U}) = \mathrm{div}\left(-\overline{p'\mathbf{u}'} + 2\mu\overline{\mathbf{u}' e_{ij}'} - \rho\tfrac{1}{2}\overline{u_i' u_i' u_j'}\right) - 2\mu\,\overline{e_{ij}' e_{ij}'} + \left(-\rho\overline{u_i' u_j'}\,E_{ij}\right), \tag{11}$$

where additional turbulent fluctuation components are unknown. Using the Boussinesq assumption, this expression can be simplified into the form:

$$\frac{\partial(\rho k)}{\partial t} + \operatorname{div}(\rho k \mathbf{U}) = \operatorname{div}\left(\frac{\mu_t}{\sigma_k} \operatorname{grad} k\right) + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \,. \tag{12}$$

Turbulent dissipation can be defined by similar equation, where the rate of dissipation per unit mass ε is defined by Eq. 13:

$$\varepsilon = 2\nu\, e_{ij}' e_{ij}' \,, \tag{13}$$

Then, the model equation for ε is derived from the equation for k, that is:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \operatorname{div}(\rho\varepsilon \mathbf{U}) = \operatorname{div}\left(\frac{\mu_t}{\sigma_\varepsilon} \operatorname{grad} \varepsilon\right) + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \,. \tag{14}$$

The main advantages of k-ε model are: the model is relatively simple; it leads to stable calculations and convergence; it allows to obtain a good prediction for many types of flow. On the other hand the disadvantages of the model are: the prediction is poor among other things for rotating flows, the flows with strong separation and certain unconfined flows; it is valid just for fully turbulent flows.

6. Comparisons of the results - experiment vs. CFD simulation

The maximum value of the suction was determined for the position MC0 and angle of roatation 45°. In Fig. 4, there is the comparison of the results – CFD simulation (Figs. 4a and 4b) and the experiment (Fig. 4c). It should be noted that pressure taps were only in the right bottom corner of the cube (see Fig. 2). So the results shown in Fig. 4 are only for this part of cube. The 3D simulation of the wind flow around the structures is shown in Fig. 4a.

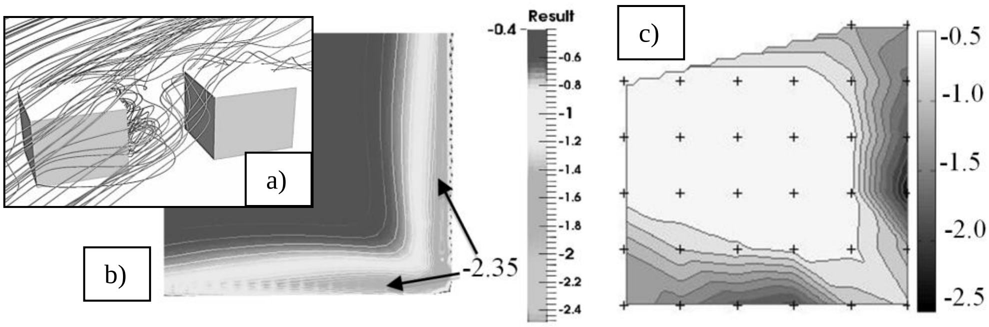

Fig. 4. Results of C_P at 45° - a) position MC1, b) CFD simulation, c) the test.

7. Conclusions

A very good coincidence between the results obtained from the experimental measurements made in BLWT tunnel and CFD simulation using k-ε model was achieved.

These results are comparable with the values mentioned in the literature and Eurocode. It can be said, that the Eurocode is conservative and pressure coefficients determined in this way are larger than the values determined by the tests and CFD simulation. In the case of self-standing cube, the results were compared with the literature [2,4]. Although, the experiment and CFD simulation did not achieve exactly the same values of suction (CFD: C_p=-2.35, the test: C_p=-2.404), deviations were relatively small and for this case used k-ε model was accurate for determination of the maximum values of pressure coefficients C_p.

Denotations of symbols

e_{ij} – component of rate of deformation tensor, [1/s],
e_{ij}' – fluctuating component of rate of deformation tensor, [1/s],
k – turbulence kinetic energy per mass unit, [m^2/s^2],
t – time, [s],
$C_{1\varepsilon}$, $C_{2\varepsilon}$ – model constants (usually equal to 1.44 and 1.92 respectively), [-],
C_p – mean pressure coefficient, [-],
E_{ij} – component of mean rate of deformation tensor, [1/s],
K – mean kinetic energy per mass unit, [m^2/s^2],
$\hat{P}_A$ – peak pressure at point A, [Pa],
$\overline{P}_A$ – mean pressure at point A, [Pa],
$\mathbf{u}'=[u',v',w']$ – fluctuating velocity vector, [m/s],
$\mathbf{U}=[U,V,W]$ – mean velocity vector, [m/s],
$\overline{U}$ – mean velocity, [m/s],
U_h – reference velocity at the top of the building, [m/s],
ε – dissipation rate of k, [m^2/s^3],
μ – viscosity, [Pa·s],
μ_t – turbulent viscosity, [Pa·s],
ν – kinematic viscosity, [m^2/s],
ν_t – kinematic turbulent viscosity, [m^2/s],
ρ – mass density, [kg/m^3],
σ_t, σ_ε – Prandtl number, [-],
τ_{ij} – component of viscous stresses tensor, [Pa].

Bibliography

[1] Tamura Y., Kareem A.: Advanced Structural Wind Engineering, Springer Japan, 2013.
[2] Richards P.J., Hoxey R.P., Short L.J.: Wind pressures on a 6m cube, Journal of Wind Engineering and Industrial Aerodynamics, 89, 2001, pp.1553-1564.
[3] Hölscher N., Niemann H.J.: Towards quality assurance for wind tunnel tests: A comparative testing program of the Wind technologische Gesellschaft, Journal of Wind Engineering and Industrial Aerodynamics, 74-76, 1998, pp.599-608.
[4] Richards, P.J., Hoxey R.P., Connel B.D., Lander D.P.: Wind-tunnel modeling of the Silsoe Cube, Journal of Wind Engineering and Industrial Aerodynamics, 95, 2007, pp.1384-1399.
[5] Wilcox D. C.: Turbulence Modeling for CFD, DCW Industries Inc., 1993.

Acknowledgements

This contribution is the result of the research supported by the Slovak Scientific Grant Agency, projects no. 1/0480/13 and no. 1/0544/15.

Summary

The influence of a nearby structure on the turbulent wind flow around the cube structure was investigated in this paper. This problem was solved by two different methods. The first one was the numerical solution by CFD simulations. Obtained results were compared with the results from tests made in the wind tunnel. Description of the problem, theory and basic equations, specification of the 3D simulations and experimental measurements were mentioned. From the evaluation of the results, pressure coefficients for this type of structure and for three solved cases, were determined. Pressure coefficients determined by the Eurocode were larger than the values determined by the tests and CFD simulations. In the case of self-standing cube, pressure coefficients were following: C_p=-2.35 (CFD), C_p=-2.404 (test). Deviations were relatively small. Hence, used k-ε model is suitable for determination of the maximum values of pressure coefficients C_p in the investigated case.