Effect of inlet conditions for numerical modelling of the urban boundary layer

Renata Gnatowska¹

¹ Faculty of Mechanical Engineering and Computer Science, Czestochowa University of Technology Al. Armii Krajowej 21, 42-200 Częstochowa, Poland e-mail: gnatowska@imc.pcz.pl

Abstract

The paper presents the numerical results obtained with the use of the ANSYS FLUENT commercial code for analysing the flow structure around two rectangular inline surface-mounted bluff bodies immersed in a boundary layer. The effects of the inflow boundary layer for the accuracy of the numerical modelling of flow field around a simple system of objects are described. The analysis was performed for two concepts. In the former case, the inlet velocity profile was defined using the power law, whereas the kinetic and dissipation energy was defined from the equations according to Richards & Hoxey [6]. In the latter case, the inlet conditions were calculated for the flow over the rough area composed of the rectangular components.

Keywords: aerodynamics of surface-mounted objects, boundary layer flows, turbulence model

1. Introduction

The application of Computational Fluid Dynamics (CFD) in the urban boundary-layer wind environment has been significantly increased in last decades [2, 4-5, 8]. Therefore a large number of codes employing a variety of solution strategies are now available. Furthermore, the atmospheric boundarylayer develops along roughened surface. Currently used of codes are based on trends in the introduction of simple alternatives to the classical logarithmic law for smooth walls that are inappropriate, even outside the detachment, and certainly wrong in the immediate vicinity of obstacles. It is known that the standard logarithmic law fails in such regions. However, the accurate reproduction of the physical nature of the boundary conditions around and on the object is not decisive for the correctness of calculations. Significant changes of parameters of the flow near the wall does not affect substantially the qualitative nature of the simulation results and only slightly impacts on the quantitative results - at least in the case of calculation prescribed. This is due to the fact that pressure and velocity fields are dominated by the large-scale movements generated by the obstacle. It is important, however, to properly model the inlet layer mundane, and it requires a careful mutual fit of model parameters of turbulence and roughness conditions in the boundary layer. Preliminary calculations without considering obstacles are necessary to validate a simulation of inlet conditions. Most of the published comparisons between calculations and experiment do not describe these types of tests. This problem becomes mainly important for uRANS or LES methods. Among the various developed techniques of modelling inlet conditions [1, 3], none of them is not easy to apply in the external flows. The evaluation of the results among various turbulence models and simulation of inlet conditions for the wind environment can benefits the selection of turbulence models and improves accuracy of simulations [7-9]. In the present study, based on the wind tunnel tests with buildings models, author investigated the performance of several RANS models, mainly including the most widely used turbulence models in industrial CFD.

This study was aimed to evaluate the improvement of modelling of the characteristics of the velocity field formed at the inlet to the calculation area and its effect on the flow around a simple arrangement of objects. The analysis was performed according to two concepts.

The first of them was definition of the velocity profile using the power law $U(z)=U_0(z/\delta)^{\alpha}$, whereas the kinetic energy (*k*) and dissipation energy (ε) was defined from the equations according to Richards & Hoxey [6]: $k=u*^2/C\mu^{0.5}$, $\varepsilon(z)=u*^2/\kappa z$, where $C_{\mu}=0.09$, κ is von Karmana constant equal 0.41 and friction velocity is $u*=0.04U_0$. The second concept assumes modelling of inlet conditions through development of the layer profile by means of the calculation of the flow over the rough area composed of the rectangular cuboid components.

2. Computational and experimental details

The geometry of the analysed case of two rectangular inline surface-mounted square cylinders immersed in a boundary layer are sketched in Figure 1. The measurements were carried out for configuration of two elements with different height, aligned in one line and the distance between them was S/B=1.5. The results presented in this work relate to a fixed ratio of object height $H_1/H_2=0.6$ and value of their "immersion" in boundary layer $H_2/\delta = 1$.

The program of the study consists of wind-tunnel experiments and numerical simulations. In the frame of the numerical study, the three-dimensional steady RANS simulations have been carried out using a commercial CFD code, FLUENT v. 17.2, with two turbulence models: RSM and k- ϵ RNG. According to the literature [2, 8-9] these models are widely used for analysis of flows for different configurations of buildings. The dimensions of the computational domain were chosen based on the best practice guidelines by Tominaga et al. [9].



Figure 1: The geometry of the analysed case

3. Selected results

Proper evaluation of the effect of the method used to generate inlet conditions requires that the analysis of the velocity field around the system of objects is made, especially before the objects and in the detachment zone behind the system of objects.

Comparison of velocity profiles obtained for selected distances before objects in comparison to the experimental profile (Figure 2) reveals their good agreement. The reference values were provided by the values of velocity U_{ref} and velocity fluctuation $U_{rms,ref}$ recorded for the height $z/H_1=1$ of the flow without objects in their location.





Figure 2: The distributions of the reduced mean velocity (a) and fluctuation velocity (b) for distances $x/B=-1.0\div-0.7$

4. Summary

No significant differences resulting from the method of inlet conditions modelling were found in the aerodynamic wake for the system of objects. Since it was necessary to model inflow conditions, especially the distribution of velocity fluctuation in the space before the object, further analysis was based on the methodology of modelling of the inflow layer by means of the initial calculations conducted over the surface with rough elements.

Numerical analysis using the *RSM* turbulence model, which takes into account the anisotropy of velocity fluctuations, should lead to more accurate mapping of the flow structure than calculations using the *k*- ε model in *RNG* version especially in the stagnation region where the assumption of isotropic turbulence is not right. For flow in the close vicinity to the ground, which is important for the wind comfort, the better efficiency representation of experimental result was obtained for the model *k*- ε *RNG*.

References

- [1] Balogh, M., Parente, A., Benocci, C., RANS simulation of ABL flow over complex terrains applying an enhanced k-e model and wall function formulation: Implementation and comparison for Fluent and OpenFOAM. J. Wind Eng. Ind. Aerodyn., 104, pp. 360-368, 2012.
- [2] Gnatowska, R., Sosnowski, M., Uruba, V., CFD modelling and PIV experimental validation of flow fields in urban environments, *E3S Web of Conferences*, 14, 01034, 2017.
- [3] Gousseau, P., Blocken, B. Van Heijst, G.J.F., Quality assessment of large-eddy simulation of wind flow around a high-rise building: validation and solution verification. *Comput. Fluids*, 79, pp. 120-133, 2013.
- [4] Lipecki, T., Flaga, A., Vortex excitation model. Part II. Application to real structures and validation, *Wind Struct.*, 16(5), pp. 477-490, 2013.
- [5] Montazeri, H., Blocken, B., CFD Simulation of Windinduced Pressure Coefficients on Buildings with and Without Balconies: Validation and Sensitivity Analysis, *Build. Environ.*, 60, 137-149, 2012.
- [6] Richards, P.J., Hoxey, R.P., & Short L.J., Wind pressures on a 6m cube; J. Wind Eng. Ind. Aerodyn., 89, pp. 1553-1564, 2001.
- [7] Richards, P.J., S. E. Norris. Appropriate boundary conditions for computational wind engineering models revisited. J. Wind Eng. Ind. Aerodyn., 99.4, pp. 257-266, 2011.
- [8] Tominaga, Y., Mochida, A., Murakami, S., Sawaki, S., Comparison of Various Revised k-ε Models and Les Applied to Flow Around a Highrise Building Model with 1:1:2 Shape Placed Within the Surface Boundary Layer, J. Wind Eng. Ind. Aerodyn., 96, 389, 2008.
- [9] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T., AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *J. Wind Eng. Ind. Aerodyn.*, 96, pp. 1749-1761, 2008.