Wind conditions in urban layout- numerical and experimental research

Marta Poćwierz[1] **and Katarzyna Zielonko - Jung**[2]

[1] *Faculty of Power and Aeronautical Engineering, Warsaw University of Technology,*
Nowowiejska 24, 00-665 Warsaw, Poland
e-mail:mpocwie@meil.pw.edu.pl

[2] *Faculty of Architecture, Warsaw University of Technology,*
Koszykowa 55, 00-659 Warsaw, Poland
e-mail: katarzyna.zielonko-jung@arch.pw.edu.pl

Abstract

This paper presents research which compares the numerical and the experimental results for different cases of airflow around a few urban layouts. The study is concerned mostly with the analysis of parameters, such as pressure and velocity fields, which are essential in the building industry. Numerical simulations have been performed by the commercial software Fluent, with the use of a few different turbulence models, including popular k-ε realizable and the Reynolds Stress Model (RSM), which is still being developed. A particular attention has been paid to accurate description of the conditions on the inlet and the selection of suitable computing grid. The pressure measurement near buildings, oil visualization and scour technique were undertaken and described accordingly.

Keywords: environmental wind engineering, airflow around buildings, CFD simulation, the k-ε realizable model of turbulence, RSM model of turbulence

1. Introduction

This paper considers one of the issues connected with CWE – wind comfort in a housing estate, which is widely described in the literature. City authorities, investors and especially all of us who live in the urban space have noticed problems, which are side effects of the wind. In some cases, the need for the aerodynamic expertise of the expanded or new buildings is required by law. Such situation arises when they are located in the wind tunnels which provide ventilation in the city. Some investors order special tests in order to improve the attractiveness of the offered estates.

The main aims of this paper are: accurate research on urban quarter; performance of numerical simulations with use of turbulence k-ε realizable model [1] and seven equations RSM model (Reynolds Stress Model) [2]; comprehensive comparison of numerical and experimental results for different layout configurations and turbulence models, as well as discussion over its usefulness in modelling the flow around buildings.

2. Model description

The subject of the study is an urban quarter in a hypothetical setting. This is an example of a typical situation when the C-shaped building is surrounded by compact buildings. The model includes two basic types of urban systems: streets and a courtyard inside the square. The dimensions and proportions of the chosen building comply with building regulations regarding issues like the right to access to daylight, mutual obscurity and fire regulations. They emulate typical conditions of densely populated districts of large cities. Two configurations were studied: the C-shaped building with opening with all buildings high equal to 18 m (Fig.1a) and the C-shaped building with opening with one of the building 2 times higher (Fig. 1b).

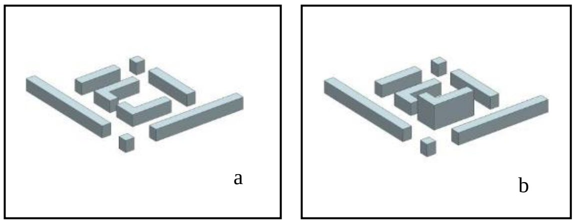


Figure 1: Studied configurations – low (a), high (b)

For the purpose of testing in the wind tunnel, the size of the buildings was scaled down, using 1:400 scale. The scaled dimensions were used for the numerical simulation.

3. Calculation model

3.1. Geometry of numerical domain

The computational domain is a rectangular prism of size 1.25m x 2.6m x 0.625m.

The height of the domain is 0.625m. It is suggested [3] that this dimension should be at least 5 times higher than the highest building.

In the case under consideration, the size of the domain on the lateral direction is equal to 1.25m and satisfies the condition described above. The other dimensions meet the criteria presented in literature [3,4].

According to the tests conducted in [5], a structural grid was used . The whole mesh consists of roughly 2.5 million elements.

3.2. Numerical calculation

The simulation was performed using FLUENT 15.0. software. Wall condition was given on surfaces of the buildings and the computational domain boundaries. The incompressible model of flow with a constant density and kinematic viscosity was applied in calculations. Steady air flow was applied, which is a common method for modelling airflow around buildings. The pressure based solver of double precision and the standard method to interpolate the pressure was used.

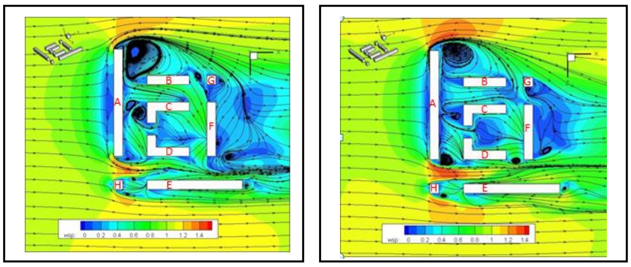
SIMPLE algorithm, which is based on the segregated method, was applied in calculations. Two models of turbulence were adopted: k-ε realizable model (RKE) and Reynolds Stress Model (RSM).

4. Experimental research

Experiments were conducted in a wind tunnel. One method was used during the research – that is measuring the pressure by a group of water gauges – and two planar methods – namely, oil visualization and sand erosion were applied. Measurement of the pressure was carried out primarily in order to compare the numerical model with the experimental one. The pressure was measured in defined points between buildings with the use of water gauges' battery connected to the model using rubber tubing. The readings were taken automatically by the system connected to the computer.

5. Comparison of numerical and experimental results

Contour maps (Fig. 2 a,b) present coefficient of the wind speed amplification for two turbulence models – RKE and RSM.



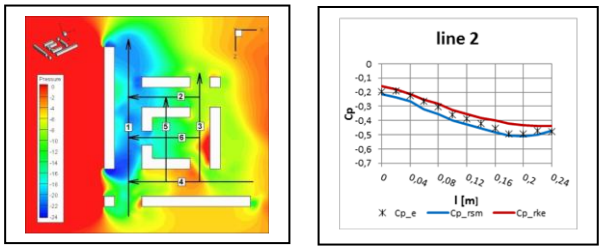
a b

Figure 2: Contour map of wind speed amplification factor with streamlines using RKE (a) and RSM (b) turbulence models

Experimental methods generate results similar in to the numerical simulations. Both turbulence models correctly represent small recirculation eddies behind buildings E, G, and H. On the other hand, bigger ones, produced by building A, cover too large area. This fact may be caused by the problems of k-ε models that have a tendency to overestimate kinetic energy of turbulence [6], which results in differences of size and location of vortexes when compared to experiments. However, using RSM did not improve this significantly. The composition of vortices and streamlines are not the same behind analysed quarter for two turbulence models. It is difficult to judge which one is more appropriate, as this is a strongly turbulised region of a complex flow.

Both models simulate a correct direction of the flow, including reversed flow between buildings B-C, C-D and in the central part of the quarter. The effect of wind nozzle between buildings A and H is modelled properly using both models of turbulence.

The following map (Fig. 3 a) presents example of pressure distribution in the analysed urban setting. The lines symbolise places, where the measurements in the wind tunnel were conducted. Next to it is the diagram of the pressure coefficient along the path nr 2 (Fig 3 b). The values of the pressure coefficient derived from numerical simulation match the ones measured



a b

Figure 3 a) Contour map of the pressure b) Pressure coefficient along line 2.

6. Summary

The obtained experimental results, supplemented by numerical calculations gave a full picture of the phenomena occurring in the tested configurations of buildings and allowed for exact interpretation. Two turbulence models were used (k-ε realizable and Reynolds Stress Model). There was no reason to use a more complex method, that is seven equations model RSM (Reynolds Stress Model). The time needed for calculation using RSM model was almost twice as long as using the popular k-ε realizable model and the results were not more accurate. In [3], it was mentioned that RSM gives more accurate results when analysing simple airflow around the block. For a more complicated geometry, the results were not improved by using RMS model.

The bigger differences in the values of obtained pressure were noticeable in the areas of strong vortexes. This exposes current problems of turbulence modelling, as RANS models seem to work worse in such cases.

References

[1] Franke,J., Hellsten, A., Schlünzen, H. and Carissimo, B., *Best practice guideline for the CFD simulation of flows in the urban environment.* University of Hamburg, Hamburg, 2007.

[2] Reiter, S., Validation Process for CFD Simulations of Wind Around Buildings, *European Built Environment CAE Conference*, 2008.

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

[4] Tominaga, Y., Mochida, A and others, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, 96, pp. 1749-1761, 2008.

[5] Gumowski, K., Olszewski, O., Poćwierz, M., Zielonko-Jung K., Comparative analysis of numerical and experimental studies of the airflow around the sample of urban development, *Bulletin of the Polish Academy of Sciences Technical Sciences*, Vol. 63, No. 3, pp. 729-737, 2015.

[6] Błazik – Borowa, E. *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).