24th International Conference
ENGINEERING MECHANICS 2018
Svratka, Czech Republic, May 14 –17, 2018
Paper **#57,** pp. 249–252, doi: 10.21495/91-8-249

CFD VALIDATION OF ABL IN SHORT WIND TUNNEL CHAMBER ALONG AN EMPTY DOMAIN

R. Gnatowska*

Abstract: *The aim of this article is numerical modeling of the atmospheric boundary layer (ABL) in a Wind Tunnel with short test section. Validation of different turbulent models through comparison with wind tunnel experiments is essential for practical applications and was examined for their suitability for the modeling of airflow in the atmospheric boundary layer. In this preliminary study, the validation of turbulent models was made with the PIV experimental measurements from a wind tunnel. It has been observed that the k-ε model is the most suitable tool for modeling the air flow in boundary-layer generated in wind tunnels (SBLWT).*

Keywords: atmospheric boundary layer, numerical simulation, turbulence models, short test section wind tunnel

1. Introduction

The importance of atmospheric boundary layer (ABL) dynamics and physics in controlling key aspects of the experimental and numerical modelling is becoming increasingly recognized. Processes such as pedestrian wind/thermal comfort, pollutant dispersion and wind-driven rain are strongly dependent on the parameters of the ABL (Jarża, and Gnatowska, 2006; Gnatowska, 2011; Gnatowska, 2015). Determination of proper scale of the natural wind structure in the wind tunnel requires an adequate roughness on the considerable length, which is possible only in tunnels with a long test-section boundary layer (Counihan, 1969; Cermak, 2003; De Bortoli, et al., 2002; Ohya, et al., 1996). Therefore advanced and original techniques for generation of the main characteristics of natural winds, as well as the formation of the atmospheric boundary layer (ABL) in short test-section boundary-layer wind tunnels (SBLWT) are required. These techniques will allow to use shorter test sections of wind tunnels for simulating the characteristics and behaviour of the atmospheric boundary layer (Barbosa, et al., 2000; Shojaee, et al., 2014). The computational fluid dynamic (CFD) simulation of the ABL has become an important research subject. CFD is a tool, which is used more and more often to study wide type of processes in the ABL, where numerical modelling is called computational wind engineering (Koblitz, et al. 2015, Gnatowska, et al. 2017). CFD seems to be the obvious way to make a new contribution to understanding the behavior of the flow fields in short test section wind tunnel. In order to understand the mechanisms of transient flow in the environment of building arrangement, a research will be conducted with the use of two parallel complementary approaches: measurements of PIV and hot-wire anemometry (HWA) and pressure in the wind tunnel, as well as numerical modeling methods.

One of the main goals of my research project will be to study and evaluate local wind comfort in built-up areas. The frame of my project “Development of a method for generation of the atmospheric ground-level layer (AGL) in the short test section of the wind tunnel” is the analysis and validation of unsteady physical phenomena. Mechanisms associated with the transient nature of air flow in atmospheric boundary layer in urban areas will be performed using RANS and LES methods. In order to achieve this goal, a method for the generation the inflow conditions in the inlet section of the wind tunnel with short test section will be developed. This type of tunnel with the multi-disciplinary feature is located in the

* Renata Gnatowska, PhD. Eng.: Institute of Thermal Machinery, Czestochowa University of Technology, Faculty of Mechanical Engineering and Computer Science, 21 Armii Krajowej Av, 42-201, Czestochowa; PL, gnatowska@imc.pcz.pl

Laboratory of Optical Measurements Methods (LOMM) of the Strata Mechanics Research Institute of the Polish Academy of sciences (SMRI PAS). In the first part of the study, the objective is to assess the airflow characteristics of the empty wind tunnel and to evaluate the boundary layer near the floor of the tunnel under such conditions and to optimize the flow in the shortest possible chamber of the wind tunnel, with the lowest pressure loss possible leading to the formation of the ABL and obtaining large Re; similar to the high values observed in urban areas. Different turbulent models with the use of ANSYS Fluent 18.1 have been examined for their relative suitability for the atmospheric boundary layer airflow.

2. Method of ABL modeling

Atmospheric boundary layer is the part of the atmosphere immediately adjacent to the earth surface. The atmospheric boundary layer the near-ground layer – inner layer, also called Prandtls' layer (Blocken, 2015; Blocken et al., 2016) features a particular sensitivity on type of terrain topography and build-up type (geometries and object configurations). These factors that depend on the friction effect cause the wind speed deficit, but also generate local complex wind phenomena. They also affect climate and wind comfort of build-up and they are significant from the several points of view. It is not only the pedestrian comfort but also the issue of health risk resulting from pollution dispersion as well as the issue originating from catastrophic events or terrorist attacks. That is why, in the last few years the analysis of the build-up areas aerodynamic with respect to pedestrian comfort becomes increasingly important as an integral part of urban planning. Basic and application aspects cause that those issues are the subject of interest for many research centres, which intensively carry out both experimental and numerical investigations.

2.1. Description of experimental method

Measurements for validation were performed using the closed circuit wind tunnel with the 1.8m long measuring section of cross sectional area 0.5 m × 0.5 m (Figure 1a). Detailed description of wind tunnel parameters can be found in Gawor et al. (2011). The velocity was specified in the wide range from 5m/s up to 50 m/s, which gives the possibility to investigate high Reynolds number cases. The measurement and test of inlet conditions and the inlet turbulence was controlled by the use of different types of special grids described in the literature (Brion, et al., 2015 and Hohman et al., 2015), which allows to generate artificially atmospheric boundary layer. The principal measurements will be obtained by PIV system; it will allow obtain both detailed description of flow-field structure and instantaneous picture of flow field. Measurements were performed using the experimental set-up comprising low speed wind tunnel and PIV system based on 5.5 Mpx sCMOS camera and double cavity, 0.2 J per impulse, 532 nm, Nd:YAG laser.

2.2. Description of numerical method

Selection of appropriate method of numerical solution is the first of all the choices of the methodology of numerical analysis. The most obvious choice is steady Reynolds-Averaged Navier Stokes (RANS), which is used most often, since Large Eddy Simulation (LES) or hybrid URANS/LES techniques require much more computational time. Opinions about the choice of the method of CFD calculations are divided, however as it comes out from most recent, detailed literature review concerning wind engineering Blocken (2015), RANS calculations are definitely most often used. Three-dimensional RANS calculations have been carried out using a commercial CFD code, FLUENT v.18.1, with the RNG k-ε turbulence model. According to the literature and previous extensive validation study (Franke et al., 2004; Montazeri et al., 2013, Gnatowska, 2018), this model is widely used for flows in a build environment. The solver was configured as pressure-based and the analysis was performed for steady state. Pressure–velocity coupling was provided by the SIMPLEC algorithm based on the relation between velocity and pressure to enforce mass conservation and to obtain the pressure field. Convergence was monitored carefully and the iterations were terminated when all residuals showed no further reduction with increasing number of iterations. The outlet of the computational domain was defined as pressure-outlet with ambient static pressure. The meshing process was performed using ANSYS Meshing 18.1. Tetrahedral elements were used to discretize the computational domain. Figure 1b shows part of the grid distribution in the test section of wind tunnel. The mesh resolution was based on the mesh sensitivity analysis and generation guidelines (Franke et al., 2004; Blocken, 2015). The mesh quality was also taken into consideration. The near-wall region of a turbulent boundary layer was considered for $y+ > 30$.

3. Results and discussion

Fourth different wind velocities in wind tunnel were tested: 5; 11; 22 and 50 m/s. Fig. 2 shows contours of longitudinal velocity in the cross-sectional plane the wind tunnel at the distance of 1,2 m from the test section inlet. The values of velocity were made dimensionless through division by the reference velocity (maximum velocity across the vertical profile). Good correlation was observed between the PIV measured and CFD velocity and turbulence intensity profiles. Although, the measured points slightly deviates away from the CFD profile very close to the wall. It is caused by PIV method.

a) b)

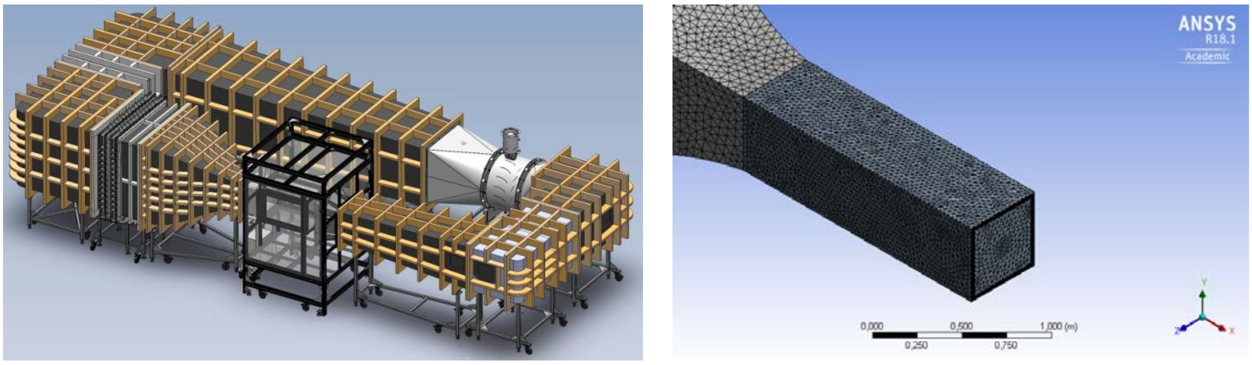

Fig. 1: Visualization of the (a) final wind tunnel design and (b) detailed CAD model of the closed-loop wind tunnel. wind tunnel.

This article shows the most suitable turbulence model - the RNG k-ε turbulence model - for modeling flow in test section boundary layer of wind tunnel. CFD results for the calculated velocity showed relatively small variations for the tested turbulence models compared with the PIV experimental data (within the range of 5–8% error). The analysis shows that the RNG k-ε model closely predicted the flow speed (3% error) and turbulence intensity (~10% error). While the Reynolds-Stress Model overpredicted the velocity results by up to 10% (Linear Pressure–Strain) and 15% (StressOmega).

a) b)

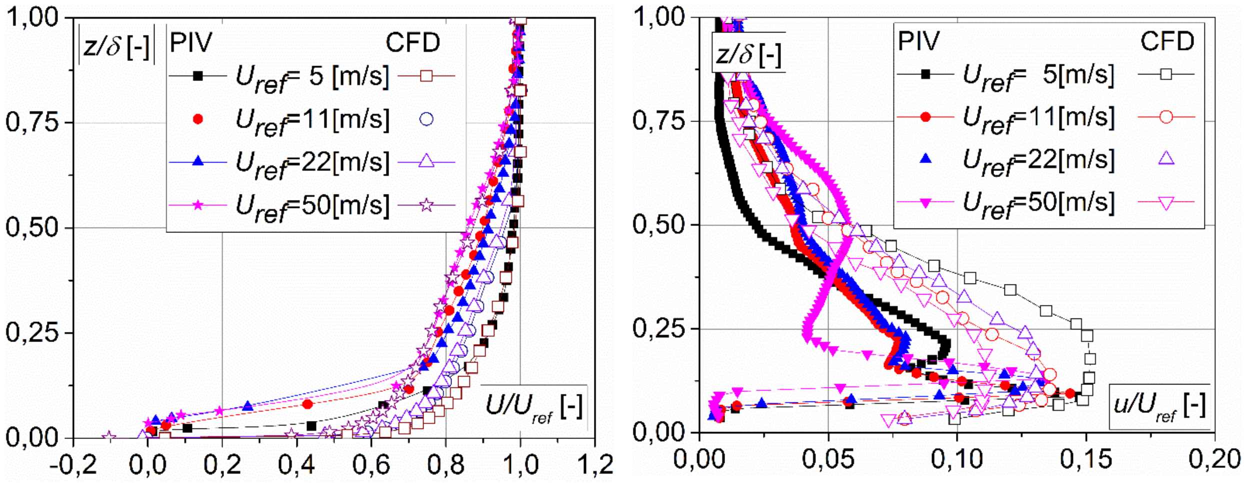

Fig. 2: Comparison between the dimensionless mean velocity and turbulence intensity of the flow at the test section for the different wind velocities in wind tunnel.

4. Conclusions

Preliminary investigation of the flow parameters in a wind tunnel was carried out by incorporating Computational Fluid Dynamics (CFD). Knowledge of velocity fields will enable optimization of flow quality in the test section of wind tunnel in the future. Furthermore, the numerical model was validated for the purpose of the empty test section. The findings from experimental PIV validation of the empty test section were found to be in good agreement with the computational results. The author compared various CFD turbulent models that were evaluated with reference to analytically and experimentally predicted velocity and turbulence values. The analysis shows that the RNG k-ε model closely predicted the test section flow velocity and turbulence intensity and was concluded as the most suitable turbulence models for this study.

Acknowledgement

This work was supported by grant 2017/01/X/ST8/00076 (National Science Centre, Poland) and statutory funds BS/PB-1-103-3011/2017/P. PL-Grid infrastructure was used to carry out the computations.

References

Barbosa, P. H., Cataldi, M. and Freire, A. S. (2000) Simulation of atmospheric boundary layer flows in short wind tunnels. In Anais do XI Congresso Brasileiro de Meteorologia, pp. 2521-2530.

Blocken, B., Stathopoulos, T. and van Beeck, J. P. A. J. (2016) Pedestrian-level wind conditions around buildings: review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment. Building & Environment 100, pp. 50-81.

Blocken, B. (2015) Computational Fluid Dynamics for Urban Physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. Building and Environment 91, pp. 219-245.

Brion, V., Lepage, A., Amosse, Y., Soulevant, D., Senecat, P., Abart J. C., Paillart, P. (2015) Generation of vertical gusts in a transonic wind tunnel. Exp. Fluids 56, 145, pp. 1-16.

Cermak, J. E. (2003) Wind-Tunnel Development and Trends in Applications to Civil Engineering. J. of Wind Engineering and Industrial Aerodynamics 91, 3, pp. 355-370.

Counihan, J. (1969) An improved method of simulating an atmospheric boundary-layer in a wind tunnel. Atmospheric Environment, 3, pp. 197-214.

De Bortoli, M. E., Natalini, B., Paluch, M. J. and Natalini, M. B. (2002) Part-depth wind tunnel simulations of the atmospheric boundary layer. Journal of Wind Engineering and Industrial Aerodynamics, 90(4-5), pp. 281-291.

Franke, J., Hirsch, C., Jensen, A. G., Krüs, H. W., Schatzmann, M., Westbury, P. S. and Wright, N. G. (2004) Recommendations on the use of CFD in wind engineering. In Cost action C14 (Vol. 14, p. C1).

Gawor, M., Gorgoń, J. and Sobczyk, J. (2011) A wind tunnel adapted to research carried out by means of the PIV method Prace Instytutu Mechaniki Górotworu PAN, 13, 1-4, pp. 257-265 (in Polish).

Gnatowska, R. (2011) Aerodynamic characteristics of three-dimensional surface-mounted objects in tandem arrangement. International Journal of Turbo & Jet-Engines, 28, 1, pp. 21-29.

Gnatowska, R. (2015) A Study of Downwash Effects on Flow and Dispersion Processes around Buildings in Tandem Arrangement Polish Journal of Environmental Studies, 24, 4, pp. 1571-1577.

Gnatowska, R. (2018) Effect of inlet conditions for numerical modelling of the urban boundary layer. In AIP Conference Proceedings AIP Publishing 1922, 1, p. 110008.

Gnatowska, R., Sosnowski, M., Uruba, V. (2017). CFD modelling and PIV experimental validation of flow fields in urban environments. In E3S Web of Conferences EDP Sciences 14, p. 01034.

Hohman, T. C., Van Buren, T., Martinelli, L. and Smits, A. J. (2015) Generating an artificially thickened boundary layer to simulate the neutral atmospheric boundary layer. Journal of Wind Engineering and Industrial Aerodynamics, 145, p. 1-16.

Jarża, A., Gnatowska R. (2006) Interference of Unsteady Phenomena in Flow Around Bluff-Bodies Arrangement. Chemical and Process Engineering, 27(3/1), pp. 721-735.

Koblitz, T., Bechmann, A., Sogachev, A., Sørensen, N. and Réthoré, P. E. (2015) Computational Fluid Dynamics model of stratified atmospheric boundary-layer flow. Wind Energy, 18, 1, pp. 75-89.

Montazeri, H., Blocken, B. (2013) CFD simulation of wind–induced pressure coefficients on buildings with and without balconies: Validation and sensitivity analysis. Building and Environment 60, pp. 137–149.

Ohya, Y., Tatsuno, M., Nakamura, Y. and Ueda, H. (1996) A Thermally Stratified Wind Tunnel for Environmental Flow Studies. J. Atmospheric Environment, 30, 16, pp. 2881-2887.

Shojaee, S. M. N., Uzol, O. and Kurç, Ö. (2014) Atmospheric boundary layer simulation in a short wind tunnel. International Journal of Environmental Science and Technology, 11, 1, 59-68.