ICWE 16
August 27-31, 2023
FLORENCE - ITALY
16th International Conference on Wind Engineering

Validation of a numerical model for the prediction of pollutant dispersion over Tokyo's Polytechnic University campus

Apostolos Krassas[1], Sergey Mijorski [2], Salvatore M. Renda [1], Eugene de Villiers[1]

[1]ENGYS Ltd., London, United Kingdom, info@engys.com
[2]SoftSim Consult Ltd., Sofia, Bulgaria, smijorski@softsimconsult.com

SUMMARY:
The aim of this study is to validate the accuracy of a Computational Fluid Dynamics (CFD) model in predicting gaseous pollutant dispersion over a large urban area. For this purpose, experimental results recently published by the Architectural Institute of Japan (AIJ) for the Tokyo's Polytechnic University (TPU) campus site (Tachibana et al., 2022) have been chosen. The set of results available include both field measurements and wind tunnel experimental data, resulting in a solid case against which the best practices and methods established in the field of Computational Wind Engineering (CWE) can be tested and validated. As part of the ongoing assessment, promising preliminary results have been obtained solving steady-state Reynolds Averaged Navier-Stokes (RANS) equations for one of the wind directions available in the dataset published by AIJ. Future work in the scope of this study will include the use of the more computationally expensive Large Eddy Simulation (LES) approach for turbulence modelling, to better capture the unsteady nature of the flow and resolve part of the turbulence spectrum more accurately. Furthermore, the assessment of all the three wind directions available in the dataset, using both full-scale and micro-scale models, and a sensitivity analysis on how the gaseous pollutant injection methodology affects the accuracy of the numerical solution, will be carried out.

Keywords: CFD, Architecture Engineering Construction, pollutant dispersion

1. INTRODUCTION

Low quality of the air breathed by human beings, caused by the ever-increasing gaseous pollutant emissions in large urban areas, poses threats to the public health and is therefore amongst the most significant problems that authorities and urbanists are faced with, on a global scale. In this context, increasing interest for accurately modelling pollutants transport and predicting their dispersion, has been raised over the past two decades and major breakthroughs have been made by professionals in the field (Blocken, 2014).

Boundary layer wind tunnels and high-fidelity CFD methodologies have both provided valuable insight into how wind flow structures develop around built-up areas and rough terrains, thus making it possible to account not only for pollutant concentration but also for other measures such as pedestrian comfort and safety factors, wind load distributions on building's façade and wind driven rain deposition. Through extensive cross comparison between CFD results and wind-tunnel experiments, the AIJ has significantly contributed to set the best practices and methodology for modelling urban microclimate physics. Part of the institute's contribution lies also in the publication of a large number of experiments, some of which also include field measurements. These experiments constitute benchmark cases that are widely used in the industry for CFD code validation, calibration, and performance enhancement. Up until now, AIJ's dispersion experiments were only conducted on simplified model-scale cases and only wind tunnel measurements were available. However, in one of their latest publications (Tachibana et al., 2022), a full-scale experiment assessing the dispersion of perfluoro in the campus of Tokyo's Polytechnic University (TPU) for three wind directions is presented. The results are validated by high-fidelity experiments conducted in an Eifel-type wind tunnel.

The scope of current work is the validation of CFD numerical models based on RANS and LES approaches against the data provided by AIJ for both wind tunnel and full-scale experiment. A single-phase, compressible, multi-component model is employed for simulating the dispersion of the gaseous pollutant into air. Contrary to the incompressible isothermal modelling assumptions used for pedestrian comfort assessment as industry standard, the more complex non-isothermal buoyant nature of the flow poses additional challenges in terms of computational cost, especially in the case of LES. Therefore, the study is also focused on assessing the trade-off between accuracy and computational cost for those methods, and possibly providing further insight in their suitability for modelling this type of applications. For the purposes of this project, a block-coupled solver, recently developed by ENGYS, is employed.

2. PRELIMINARY SIMULATION STUDY

As part of preliminary work, a steady-state RANS approach was employed to model the wind tunnel experiment for a single wind direction, namely 180^o.

2.1. Computational Domain and Mesh

The computational domain has a cross-section of 1.2m x 1m in width and height respectively, which is in line with the dimensions of the test section in the actual wind-tunnel used for the experiment. The domain was extended by 5H upstream and more than 15H downstream of the target area to be compliant with the AIJ and COST guidelines (Franke et al. 2011). H denominates the height of the campus' tallest building. A hex-dominant computational grid with a coarsest mesh size of 0.052m was created in HELYX. Additional refinements were introduced in a gradually increasing fashion for cells located below 0.4m in the direction normal to the ground. A total of 7 refinement levels (each refinement halving the base mesh) and 5 boundary layer prisms were used, resulting to an approximately 32 million cells mesh (see Figure 1).

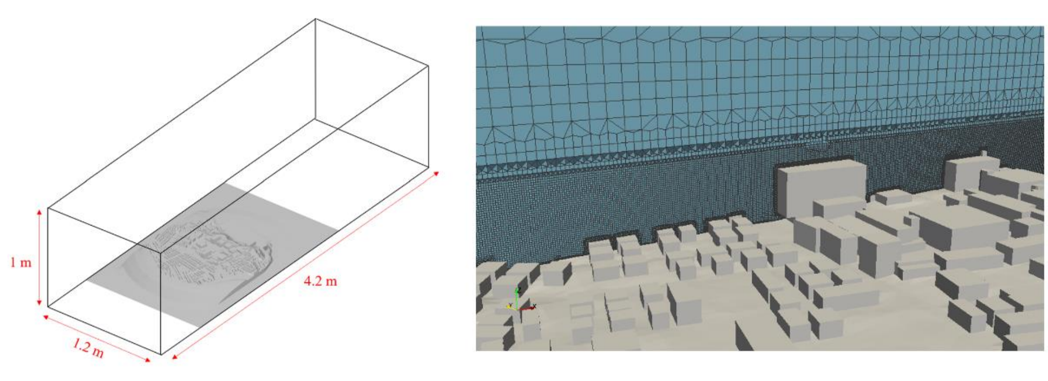

Figure 1. Numerical wind tunnel set up and spatial discretization.

2.2. Settings

The k-Omega SST turbulence model was employed for turbulence closure in the context of RANS modelling. The inflow boundary conditions were determined based upon the experimental data provided by AIJ including velocity components and fluctuations measured in the empty wind tunnel. Two separate approaches were used for treating the top and lateral boundary patches: a) smooth non-slip walls to replicate the effect of the actual walls of the wind tunnel and b) slip walls to model the flow conditions of the field experiment. The floor of the wind tunnel and the campus terrain were modelled as rough walls (Blocken et al., 2007).

3. PRELIMINARY RESULTS

The results of the preliminary study are presented in this section.

3.1. Dimensionless Concentration

The dimensionless concentration C* defined by equations (1) and (2) was used as a quantitative measure for comparing results. In the equations, C_0 is the reference concentration (in ppm), C is the concentration at each measurement location (in ppm), q is the tracer gas volumetric flow rate (expressed in m^3/s), H and U_H are the reference height (in m) and reference wind speed (in m/s), respectively.

$$C_0 = \frac{q}{U_H H^2} \tag{1}$$

$$C^* = \frac{C}{C_0} \tag{2}$$

Figure 2 shows a comparison between the dimensionless concentration predicted by the two CFD simulations and the values measured in the field and wind tunnel experiments.

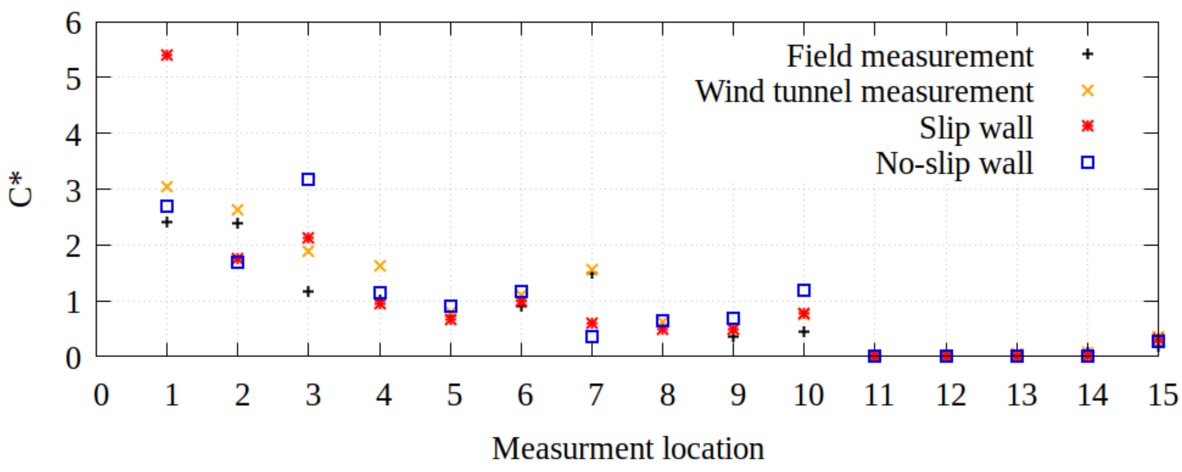

Figure 2. Dimensionless concentration on measurement locations.

4. CONCLUSIONS & FURTHER WORK

The preliminary results presented in Section 3 give us a ballpark figure of the extent at which a steady-state RANS simulation can accurately predict pollutant dispersion over a built-up area. In particular:

- The estimate of the overall dispersion trend is in good agreement with both field and wind tunnel measurements.
- The deviation is higher for points in proximity of the gas source (i.e., 1, 2, 3 and 10 in Figure 2) while gas concentration at remaining locations far from the injection point is more accurately predicted. This suggests sensitivity to the way the pollutant is injected in the computational domain and to the unsteady nature of the flow requiring more accurate resolving of the turbulence spectrum and diffusion process.

As part of the further work within the scope of this project:

- Unsteady RANS and LES simulations will be carried out and compared with the steady-state RANS approach both in terms of accuracy and computational cost. This comparison aims to offer an overview of the trade-off between prediction accuracy and turnaround times when it comes to model pollutant dispersion physics.
- Two additional wind directions will be investigated.
- A sensitivity analysis will be performed to determine how the pollutant injection method inside the fluid domain affects the overall accuracy of the numerical predictions.
- The results between a micro- and a full-scale model will be cross compared to determine how model size affects the prediction accuracy.

ACKNOWLEDGEMENTS

This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951745. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Italy, Slovenia, France, Spain.

REFERENCES

Architectural Institute of Japan, 2007. Guidebook for Practical Applications of CFD to Pedestrian Wind Environment around Buildings. Architectural Institute of Japan.

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

Blocken B. 2014. 50 years of Computational Wind Engineering: Past, present and future. Journal of Wind Engineering and Industrial Aerodynamics.

Franke, Jörg, Hellsten, Antti & Schlünzen, Heinke & Carissimo, Bertrand. 2011. The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: A summary. International Journal of Environment and Pollution. 44. 419-427. 10.1504/IJEP.2011.038443.

Tachibana, T., Yoshie, R., Nakayama, S., Kishida, T., Miyashita, K., Sasaki, R., 2022. Comparison between field measurement and wind tunnel experiments of gas dispersion in an urban area and verification of similarity law. Journal of Wind Engineering, 47(3), 39-52.

Tamura, T., Nozawa, K., Kondo, K., 2008. AIJ guide for numerical prediction of wind loads on buildings. Architectural Institute of Japan.