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

CFD-based investigation of air quality in urban areas: A case study in Brussels, Belgium.

S.K. Raghunathan Srikumar [1], G. Mosca[1], L. Cotteleer[2], A. Parente[2], A. Gambale[1]

[1]BuildWind SPRL, Rue Bara 175, 1070 Brussels, Belgium, skrsrikumar@buildwind.net
[2] Université Libre de Bruxelles, Brussels, Belgium, leo.cotteleer@ulb.be

SUMMARY:
Pollutant dispersion in a neighbourhood around the ULB Solbosch University campus, Brussels, was investigated using Computational Fluid Dynamics (CFD) simulations. All the buildings and terrain shapes within a circular region of radius 1.3 km were explicitly modelled in the computational mesh. Steady, incompressible RANS based CFD simulations were performed for the wind using a modified k-ω SST turbulence model, implemented with an improved Atmospheric Boundary Layer (ABL) approach. For pollutant dispersion, passive scalar simulations were performed on top of the wind simulations using a variable Schmidt number based framework. The CFD setup was first validated with a well-established wind tunnel test case from the Architectural Institute of Japan. The validated setup was then used to perform multiple wind simulations on the university campus, followed by passive scalar simulations for pollutant dispersion. Finally, CFD results were compared with measurements obtained from sensors placed around the campus for validation of the CFD model.

Keywords: Pollutant dispersion, air quality, wind simulation

1. MOTIVATION

Air pollution is a major concern in many cities around the world, with negative impacts on human health, the environment, and the economy. Pollutants from cars, factories, and household heating systems are dispersed in the atmosphere and can impact air quality over a wide area. Studying the dispersion of pollutants in cities is therefore crucial to understand the impact of air pollution and develop strategies for mitigating its negative effects. Computational Fluid Dynamics (CFD) simulation is a powerful tool for discerning the complex flow patterns of wind and pollutants around buildings in a city (Blocken et al., 2013; Tominaga et al., 2013). By modelling the dispersion of pollutants, valuable insights into how they are transported by convection and diffusion through the urban environment can be gained. This information can be used to develop strategies for mitigating air pollution and improving air quality in cities. In addition to its potential applications in urban planning and environmental engineering, CFD simulation can also provide important information for public officials and policymakers to make informed decisions about how to protect the health of city residents and reduce the negative impact of air pollution on communities. Furthermore, the use of CFD simulation can help to identify hot spots of air pollution within a city, allowing targeted interventions to be implemented in these areas to improve air quality.

In the current work, a CFD analysis of pollutant dispersion was performed on a neighbourhood around the Solbosch University campus in Brussels. Buildings enclosed within a circular area of radius = 1.3 km were explicitly modelled in the computational mesh. Within this region, dispersion of different pollutant sources such as NO_2, $PM_{2.5}$, PM_{10}, etc were studied in detail.

2. METHODOLOGY

All the analyses in the current work were conducted using the open source FVM-based solver OpenFOAM version 7. Wind simulations were performed first, followed by passive scalar simulations of different pollutants. The pollutant source strengths and locations were estimated from the traffic data provided by the Brussels Mobility and the sensor measurements (QSENSE-Air sensors from Macq) in and around the campus. These were then used to set the appropriate boundary conditions for the pollutant dispersion simulations. For the wind simulation, an improved Atmospheric Boundary Layer (ABL) framework implemented with a modified k-ω SST RANS model (Bellegoni et al., 2022; Parente et al., 2011; Longo et al., 2017) was used. This particular model was chosen because it alleviates the well-established issue of horizontal inhomogeneity of velocity profiles in an ABL simulation (Blocken et al., 2007). For the pollutant dispersion simulation, correct specification of the turbulent Schmidt number (Sc_t) is essential to get an accurate concentration field (Longo et al., 2020; Lauriks et al., 2021). In this regard, a variable Sc_t formulation was adopted based on the works of Longo *et al* obtained from Manifold-Generated Principal Component Analysis (MG-PCA) of dynamic DES data (Longo et al., 2020). This variable Sc_t is given as a function of molecular Schmidt number (Sc), turbulent Reynolds number (Re_{turb}), vorticity (S) and strain rate (Ω) tensors and is reported to have performed much better than the standard RANS approaches (Longo et al., 2020).

$$Sc_t = exp\left(a\,Sc - bRe_{turb}^{c} - d\,S - e\,\Omega\right) \tag{1}$$

where a, b, c, d, e are constants with values *0.6617, 0.8188, 0.01, 0.031, 0.0329* respectively (Longo et al., 2020).

3. RESULTS

The CFD setup discussed above was been validated on the well established cube array test case from Architectural Institute of Japan (AIJ - Case K), given in terms of non-dimensional concentration K, defined as :

$$K = \left(\frac{\frac{C_m}{C_s}U_{ref}H^2}{Q_s}\right) \tag{2}$$

where C_m is the measured tracer concentration in *ppm*, C_s denotes source tracer concentration in *ppm*, U_{ref} specifies the reference wind speed in *m/s* measured at H_{ref}, U_{ref} is the model building height H and Q_s denotes the total source strength/source flow rate in m^3/s. One of the validation results is shown below in Fig. 1:

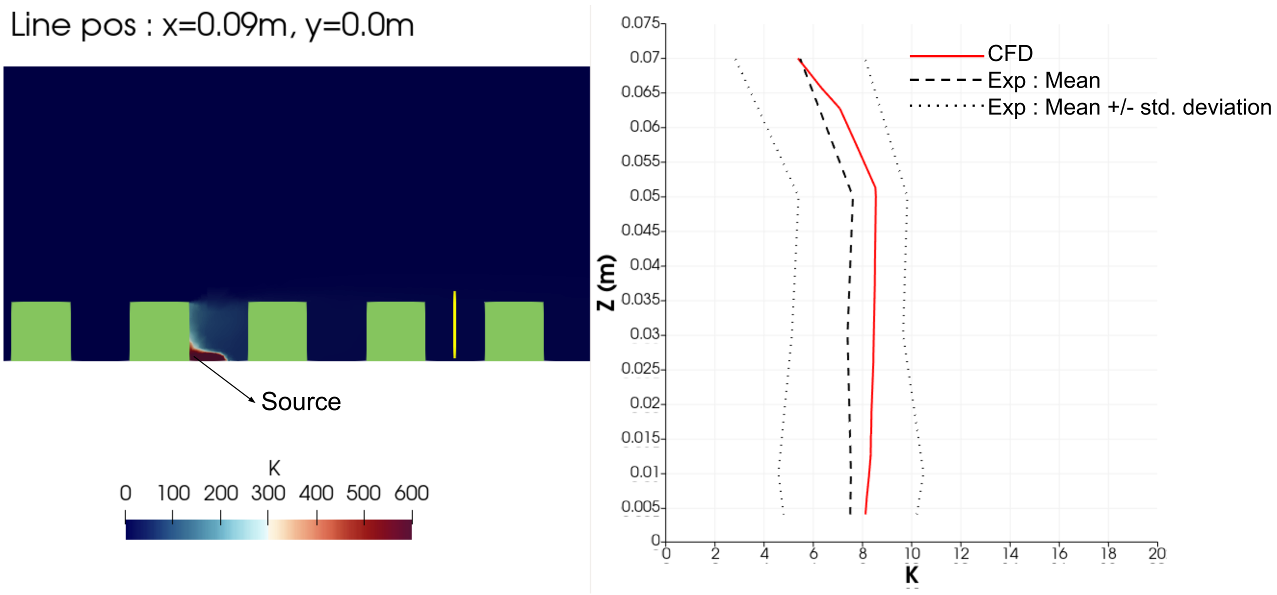

Figure 1. Validation of non-dimensional concentrations K with experimental data from AIJ cube array case

The figure shows a slice in the middle of the cube array plotted with contours of non-dimensional pollutant concentration *K* obtained from CFD simulation. Exit concentration of the gas emitted from the source location is 10^6 *ppm*, which is emitted at a flow rate of $3.6 * 10^{-6}\ m^3/s$. *K* values are compared on a line at the location shown in Fig. 1 between CFD and measurements. Values obtained from CFD fall within 1 standard deviation from the mean experimental values.

The validated CFD setup was then applied to the campus area through multiple wind simulations (an example is shown in Fig. 2) corresponding to the most frequent wind directions as measured at the nearest meteorological station, followed by the pollutant dispersion simulations. The pollutant concentration results were then compared to the sensor data obtained from the measurement campaign to gauge the accuracy of the CFD setup on a realistic urban scale.

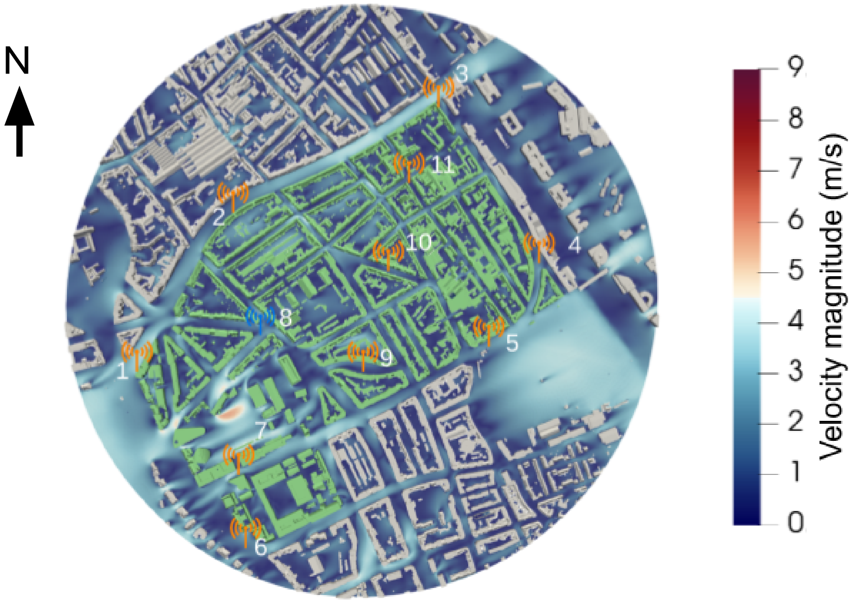

Figure 2. Velocity contours at 2m from the ground around the Solbosch campus. Wind blowing from the South-West direction. Sensor locations around the campus are marked

ACKNOWLEDGEMENTS

The authors gratefully acknowledge Innoviris and the Brussels Capital Region for co-funding this research in the context of the project SPICECO, under grant 2021-RDIR-17A.

REFERENCES

Blocken, B., Tominaga, Y. and Stathopoulos, T., 2013. CFD simulation of micro-scale pollutant dispersion in the built environment. Building and Environment, 64, pp.225-230.

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

Bellegoni, M., Cotteleer, L., Srikumar, S.K.R., Mosca, G., Gambale, A., Tognotti, L., Galletti, C. and Parente, A., 2023. An extended SST k− ω framework for the RANS simulation of the neutral Atmospheric Boundary Layer. Environmental Modelling & Software, 160, p.105583.

Hargreaves, D.M. and Wright, N.G., 2007. On the use of the k–ε model in commercial CFD software to model the neutral atmospheric boundary layer. Journal of wind engineering and industrial aerodynamics, 95(5), pp.355-369.

Lauriks, T., Longo, R., Baetens, D., Derudi, M., Parente, A., Bellemans, A., Van Beeck, J. and Denys, S., 2021. Application of improved CFD modeling for prediction and mitigation of traffic-related air pollution hotspots in a realistic urban street. Atmospheric Environment, 246, p.118127.

Longo, R., Ferrarotti, M., Sánchez, C.G., Derudi, M. and Parente, A., 2017. Advanced turbulence models and boundary conditions for flows around different configurations of ground-mounted buildings. Journal of Wind Engineering and Industrial Aerodynamics, 167, pp.160-182.

Longo, R., Bellemans, A., Derudi, M. and Parente, A., 2020. A multi-fidelity framework for the estimation of the turbulent Schmidt number in the simulation of atmospheric dispersion. Building and Environment, 185, p.107066.

Parente, A., Gorlé, C., Van Beeck, J. and Benocci, C., 2011. Improved k–ε model and wall function formulation for the RANS simulation of ABL flows. Journal of wind engineering and industrial aerodynamics, 99(4), pp.267-278.

Tominaga, Y. and Stathopoulos, T., 2013. CFD simulation of near-field pollutant dispersion in the urban environment: A review of current modeling techniques. Atmospheric Environment, 79, pp.716-730.