LEAF

Large-eddy simulation study on the effect of vehicle-induced turbulence and exhaust fumes on wind flow and pollutant dispersion in urban street canyons

B. Maronga, G. Motisi*, Institute of Meteorology and Climatology, University of Hanover*

In Short

- Vehicle-induced turbulence and exhaust fumes
- Urban street canyon
- The PALM model system
- Incorporation of moving vehicles and validation

The number of people in urban environments is continously increasing. In 2018, approximately 4.2 billion people, which is 55.3 % of the worlds population, lived inside urban settlements [1]. As urbanization is expected to continue, its environmental impact is of growing interest.

A multitude of research studies already focused on the wind flow and the transport of pollutants in urban street canyons. Apart from the ambient wind, the presence of vegetation, possible thermal effects etc., another important factor has become known to be critical: the vehicle-induced turbulence (VIT) and exhaust fumes interacting with the turbulent flow [2,3].

Common tools for the investigation of the processes within urban street canyons are either field experiments, wind tunnel studies or computational fluid dynamic (CFD) models. Up to now, mainly the Reynolds-Averaged-Navier-Stokes (RANS) technique was applied for CFD studies, whereas nowadays, thanks to increasing computational power, the Large-Eddy simulation (LES) technique became an useful research tool. An incorporation of vehicles and VIE in an environmental LES setup, as proposed in the present study, has not yet been carried out successfully. On these grounds, the implementation of moving objects into turbulence-resolving models as well as the effect of VIE on the turbulent flow in urban street canyons including its interaction with the most important processes shall be tackled in the proposed project. For its execution the PALM model system shall be used.

In the first project phase the validation study for the static case was performed. For this purpose, a Vauxhall AstraVan was mapped in a virtual wind tunnel environment analogous to the wind tunnel experiments according to Carpentieri et. al. (2012) [4]. Here, we applied two different methods based on a solid obstacle (topography method) and a newly-developed so-called air block method. A sensitivity study was conducted with successively decreasing grid spacing in order to determine the required resolution for resolving the flow around the obstacles properly. Previous LES of the flow around simple geometries like cube-like buildings showed that a face of such a building should be resolved by at least 32 grid points (e.g. Letzel al. 2008 [5]), which was also confirmed in the present case and resulted in a grid spacing of 0.05 m. The figure below (Fig. 1) shows selected vertical profiles of the longitudinal dimensionless mean speed (u/u_{ref}) along the vehicle centre line (y=0) for the air-block method with a grid spacing of 0.05 m, the topography method with a grid spacing of 0.05 m and the 1:5 model in the wind tunnel experiments. The x-axis shows the dimensionless ratio of u to u_{ref}, whereas the y-axis represents the normalized height z/h. The longitudinal velocity deficit in the wake is clearly visible in all cases. In the far wake (x/h = 3.80 and x/h = 7.47), the topography method and the air block method show hardly any differences. This area is well represented by both methods compared to the experimental data. At near wake (x/h = 0.40 to x/h = 2.13), the topography method shows a better agreement with the experimental data than the air-block method, which, however, was predictable due to the simpler technical basis.

A consistent picture emerged throughout the study, from which it can be concluded that the air block method is very well suited for representing vehicles. Compared to the topography method, it has weaknesses near the ground and in the increased friction, but scores in its simple implementation. It can be assumed that the same picture is also seen for the moving case.

The planned simulations to be conducted will focus on the validation of the air block method for the moving case and will consist of three parts. First, the simulations will be performed in a wind-tunnel-like setup described in [2]. In comparison with the respective experimental data it will be proven that the newly-developed method can accurately reproduce these data and is thus equally robust and suitable in representing not just resting but also moving vehicles in PALM. The second part will focus on the effect of the position of the emission source. For this purpose, the release of pollutants will be represented by stationary line sources as well as by point sources at the typical location of exhaust pipes attached to the individual vehicles. The data will be compared with previous RANS model results outlined in Gross

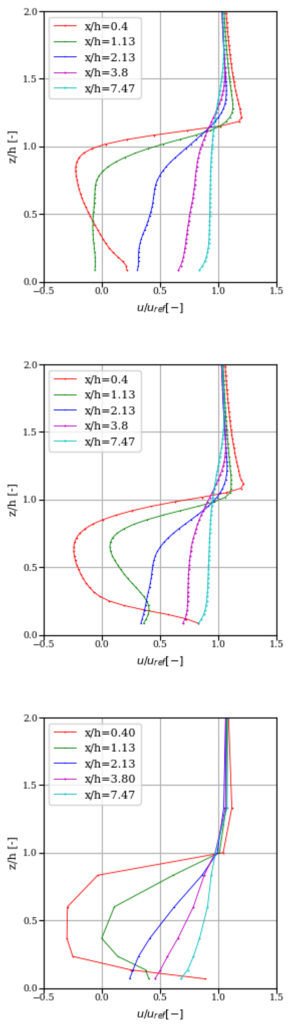

***Figure 1:** Selected vertical profiles of longitudinal dimensionless mean speed (u/u_{ref}) along the vehicle centre line (y=0) for the air-block method (top), the topography method (middle) and the 1:5 model in the wind tunnel experiments (bottom)*

(2016) [6] in order to assess whether the line source concept is an adequate represenation of vehicular emissions in LES models. Last, different free-stream wind velocities and directions will be prescribed in order to investigate the interaction between the vehicle induced effects and the dynamically-induced canyon flow and transport of pollutants. By doing so, information on the effect of a more realistic traffic situation can be obtained and compared against idealized configurations.

The planned simulations require enormous computational ressources. This is partly due to the fact that a realistic representation of a vehicle with all its flow-influencing contours requires a correspondingly high resolution, i.e. small grid spacing. On the other hand, the time step is limited by both the fine grid spacing (due to the CFL-Criterion) and by the high flow velocities that also have to comply the CFL criterion. Moreover, environmental LES has to cover all relevant scales of atmospheric turbulence so that larger model domains have to be used than in traditional engineering LES setups. For this purpose a significant number of processor cores, memory and computing time is required, which in turn can only be satisfied by massively parallel computer architectures such as the HLRN-IV system.

WWW

https://www.meteo.uni-hannover.de/en/

More Information

[1] United Nations, Department of Economic and Social Affairs, Population Division, https://population.un.org/wup/Publications/Files/WUP2018-Report.pdf

[2] Kastner-Klein, P., Fedorovich, E., Rotach, M., *Journal of Wind Engineering Industrial Aerodynamics* **89**, (2001). doi:10.1016/S0167-6105(01)00074-5

[3] Solazzo, E., Britter, R., *Boundary-Layer Meteorology* **124**, (2007). doi:10.1007/s10546-007-9176-7

[4] Carpentieri, M., Kumar, P., Robins, A., *Atmospheric Environment* **62**, (2012). doi:10.1016/j.atmosenv.2012.08.019

[5] Letzel, M.O., Krane, M., Raasch, S., *Atmospheric Environment* **42**, (2008). doi:10.1016/j.atmosenv.2008.08.001

[6] Gross, G., *Meteorologische Zeitschrift* **25**, (2016). doi:10.1127/metz/2016/0797

Project Partners

Prof. Prashant Kumar, University of Surrey; Dr. Matteo Carpentieri, University of Surrey; Prof. Petra Klein, University of Oklahoma

Funding

DFG MA 6383/3-1

DFG Subject Area

313-01