17th International Conference on
Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes
9-12 May 2016, Budapest, Hungary

SIMULATIONS OF TRAFFIC RELATED POLLUTANTS IN A MAIN STREET OF RIO DE JANEIRO CITY (BRAZIL) USING COMPUTATIONAL FLUID DYNAMICS MODELLING

Roseane A.S. Albani [1] *Beatrice Pulvirenti* [2] *and Silvana di Sabatino* [1]

[1]Dipartimento di Fisica e Astronomia ,Università degli Studi di Bologna, Bologna, Italy
[2]Dipartimento di Ingegneria Energetica, Università degli Studi di Bologna, Bologna, Italy

Abstract: In this work we study atmospheric pollutant dispersion associated to traffic emissions in a main street of down town Rio de Janeiro (Brazil) using Computational Fluid Dynamic (CFD) modelling. Our study area considers the "Presidente Vargas" Avenue, one of the most important artery in down town Rio de Janeiro, having a traffic count of about two hundred thousand vehicles every day. This artery forms in some part a street canyon in a classical sense while the remaining is embedded in more complex urban structures. The study area is a region nearby "Central do Brasil", which is a terminal railway station from where approximately six hundred thousand people make their way daily. The performance of CFD methodologies based on both Reynolds-averaged Navier-Stokes (RANS) and a large-eddy (LES) simulations for carbon monoxide (CO) dispersion is investigated. Numerical results are compared with real concentration data from one monitoring station to highlight the differences between RANS and LES approaches.

Key words: *Computational Fluid Dynamics, Atmospheric Dispersion***,** *urban areas*

INTRODUCTION

Atmospheric dispersion continues to be an active research topic given its relevance in the assessment of air pollution impact on people's health. Traffic emissions are the main source of pollution in urban areas, usually located among or around buildings. Pollutant sources are immersed in the very complicated flow patterns generated by the interaction between airflow and buildings leading to a high complex concentration field whose prediction remains challenging for most current numerical models.

Details of the flow around buildings can be investigated using computational fluid dynamics (CFD) approaches, which has been extensively used in simulations of dispersion phenomena in urban areas during this last decade as largely documented by Anand et al. (2016), Santiago et al. (2010), Di Sabatino et al. (2008) and many others. Despite the several advantages, CFD models have large computational costs still restricting its range of application to small domains (few hundreds to 1 km). In addition, CFD models needs to be thoroughly set up and evaluated case by case, given the large number of parameters to be adjusted to be successful in real scenarios. New simulations requiring different set up to account for real urban geometries as well as specific environmental physical conditions, still depend upon careful evaluation against concentration and meteorological data measured at the site.

The aim of this work is to investigate the atmospheric dispersion considering the complex scenario of Rio de Janeiro down town, which has a highly heterogeneous distribution of the buildings, besides being influenced by the Guanabara bay breeze (on the North direction), just as the sea Breeze (in the South direction) in the morning and afternoon-night periods, respectively. Although many studies focus on evaluating pollutant concentration in rather idealized street canyons here we focus on real scenarios using both Reynolds averaged Navier-Stokes (RANS) and Large-Eddy (LES) simulations to document their performance.

DESCRIPTION OF THE STUDY SITE

The region of study encompass part of the President Vargas avenue, one of the main streets of Rio de Janeiro. The selected built area covers a surface of ~ 657000 m2, which includes a very complex geometry, with asymmetric buildings of different heights, ranging from 10 to 80m and different spacing among them. Trees of different species are spread out over the whole site. The area is bounded from north/east by the Guanabara bay (~1200 m), and from the south by the Atlantic ocean (~7500 m). Near surface wind is influenced by both bay and sea breezes as reported by Pimentel et al. (2014). Figure 1 (right) shows the computational domain (inside the yellow line) where the air quality station is located. The meteorological data for the inlet conditions were obtained in the Santos Dumont airport (left).

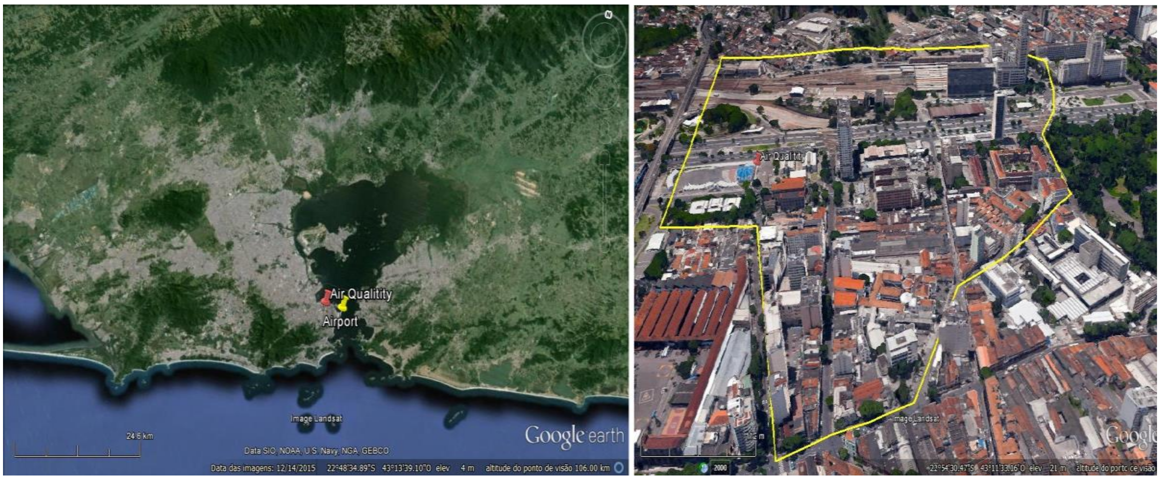


Figure 1. The site of studying encircled by the yellow line. Source: Google earth.

METHODOLOGY

We used meteorological and air quality data provided by the INEA (Instituto Estadual do Ambiente do Rio de Janeiro) and METAR data obtained from the Santos Dumont Airport (http://www.redemet.aer.mil.br). The meteorological and concentration data were taken considering the day 23/07/2013 at 6 PM, where neutral conditions could be found. To estimate the emissions, we used the curve of the volume of vehicles along a day, considering streets around the President Vargas avenue, as shown in Figure 2.

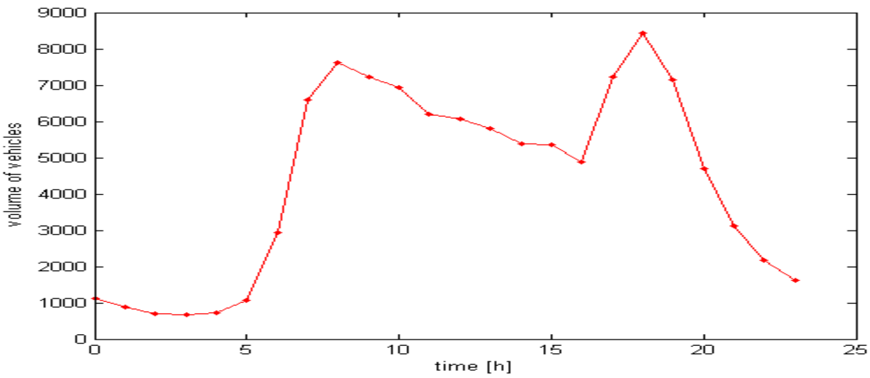


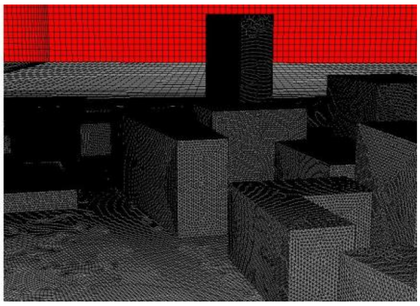
Figure 2. Volume of vehicles along the day 23/07/2013 nearby the air quality station.

Detailed information about the number of vehicles used in this study were publicly available (http://www.denatran.gov.br/frota2013.htm) while estimate of hourly emissions were obtained by using the EMIT software kindly made available by CERC Lts according to vehicle category and typical traffic speed.

SET UP AND CFD SIMULATIONS

Simulations of pollutant dispersion under neutral conditions were performed by means of Fluent ANSYS 17.0. Both RANS (Reynolds Averaged Navier-Stokes equations) and LES (Large Eddy Simulations) methodologies were employed to simulate dispersion of carbon monoxide (CO) in Rio de Janeiro down town. A description of the set of equations of the RANS and LES models can be found for example in Santiago et al. (2010). The urban boundary layer height was estimated as $\delta = 5H$, where H is the average building height while the usual equilibrium profiles of wind speed, turbulent kinetic energy (TKE) and dissipation rate were used as inlet conditions (see e.g. Di Sabatino et al. 2008). The friction velocity was estimated via the log-law expression with wind speed measured at 10 m height at the airport Santos Dumont (outside the domain). The computational domain, shown in Figure 3, is a volume of 3L x 9L with a height of 3L, with L the dimension of the area of interest considered in Figure 1. We used symmetry boundary conditions at the top of the domain and pressure-outlet conditions in the outflow.

MESH CONSTRUCTION
Mesh building was a rather important and challenging phase especially for such complex structures as in our case. In general terms, the entire domain used for simulations is larger than the volume of interest. Such strategy is employed to ensure that some constraints, imposed by the boundary conditions will not affect excessively the solution in the built area.



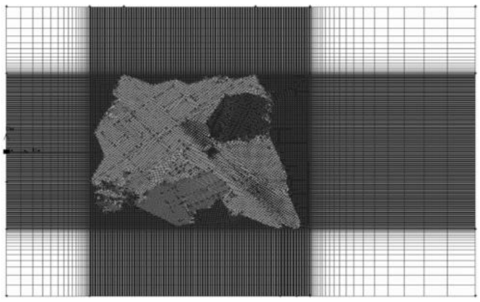


Figure 3. Structure of the computational domain in the built area (left) and the whole domain (right).

The whole domain was split into several sub domains of different sizes in order to control the mesh building process. A combination of structured and unstructured meshes composed by hexahedral and tetrahedral elements was designed since it allows a greater flexibility, accuracy and computational efficiency, compared to a purely structured or unstructured mesh. In the free stream region a structured mesh was created, since in such region the flow is predictable and stable. On the other hand, an unstructured mesh was generated for the build region, where the flow pattern is unknown. The grid is built by sweeping a surface ground mesh three times resulting in three different resolution layers. In the first layer near the ground level, a finer mesh is employed to obtain a better description of the flow and concentration at pedestrian level. Considering the second layer, a lower resolution is required compared to the ground level layer, since into this part of the domain large flow structures are expected. The third is the roughest layer, since no high resolution is required there. The mesh was constructed with 2-4-9 million of elements. The finer resolution of the element in the x, y and z direction was 0.5 m, based on convergence analysis.

RESULTS AND DISCUSSION
Two different models have been compared: k-epsilon and LES. For the k-epsilon model, standard wall functions and constants have been used, with a standard choice of the turbulent parameters. For the LES model, a fixed time step of 0.1 s was chosen, with a second order implicit discretization. The flow field obtained with the two turbulence models shows channeling and vortices with the same shape and dimensions, as shown in Figure 4.

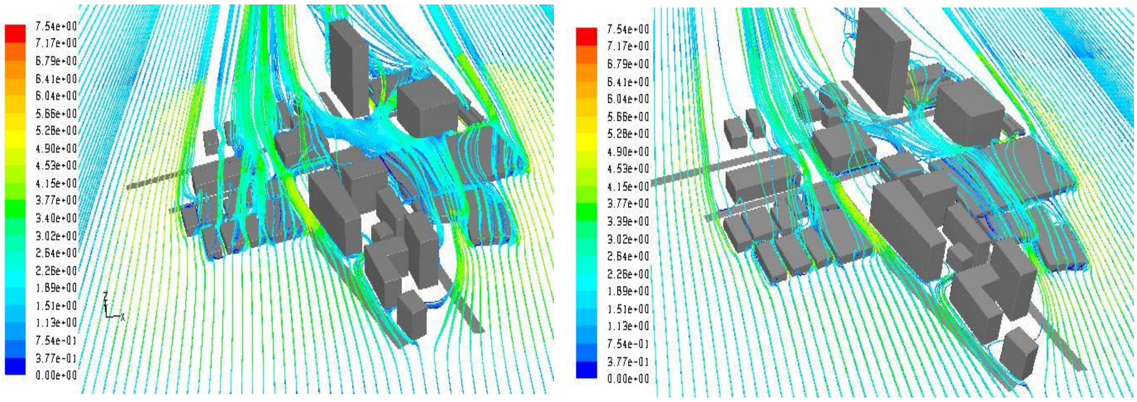


Figure 4. Comparison of the streamlines obtained over the built area with the k-epsilon model (left) and LES (right).

The flow channels mainly in the two roads: “Rua de Santana” (shown in figure 5) and “Rua General Caldwell”, as their direction is aligned with the direction of the flow. Then, vortices are observed in many areas near the streets, where large buildings confine the secondary flows. Figure 5 shows that the main vortices obtained by the two turbulence models are similar. However, small vortices are captured only by the LES model.

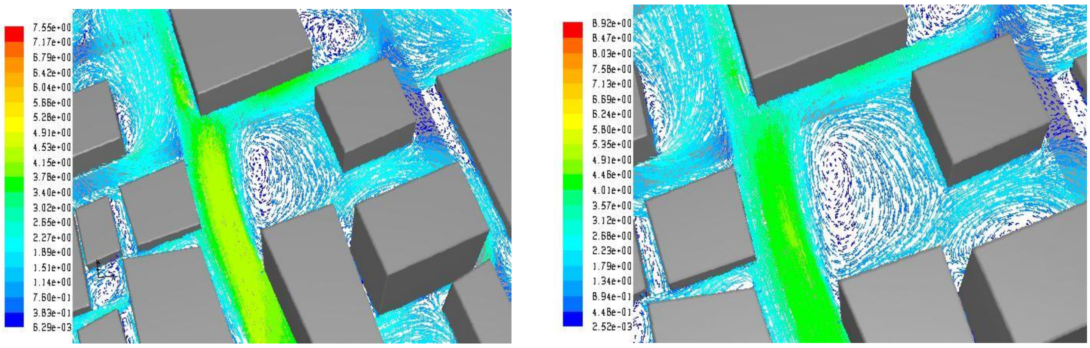


Figure 5. The vectors field in the area.

Concentration of CO does not shows high values in such recirculation zones, as the main streams carry most of the pollutant and in these areas CO is diluted. The validation has been obtained by a comparison with concentration data obtained from the air quality station positioned in the point remarked by a yellow triangle in the Figure 6 (left). Figure 7 shows CO concentration as a function of vertical direction, in the point where the station is located (yellow triangle).

The red line represents simulation result with the k-epsilon model, while the blue line corresponds to the results obtained with the LES model. The measured value in z = 0 m is 300mg/m^3. Figure 7 shows that both numerical results are lower than the CO measured data, with the results obtained by RANS model closer to the measurements than the ones obtained by LES. The reason of the under prediction could be related to the fact that the measured data are averaged over the whole day, while numerical results are instantaneous and related to the case of wind coming from South.

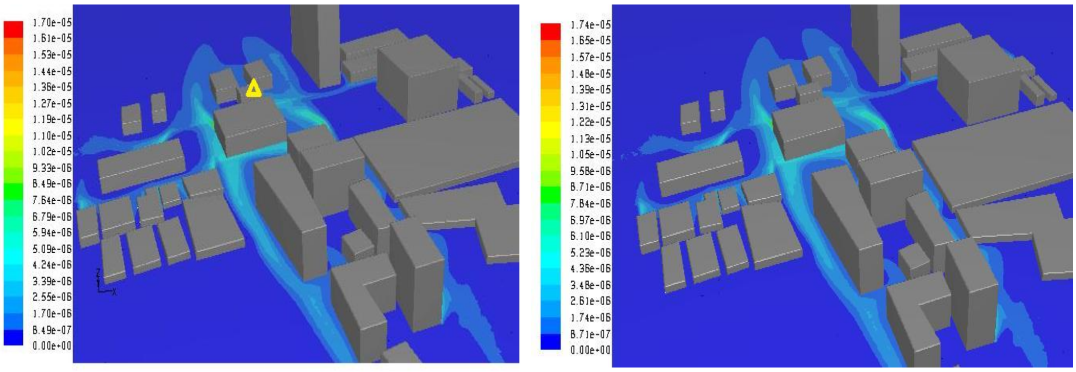


Figure 6. CO concentrations at a plane at z=2 m, for the case with k-epsilon model (left) and LES (right).

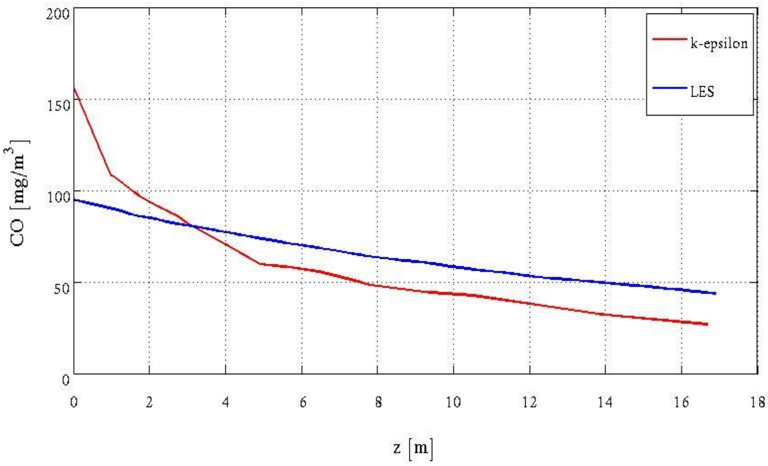


Figure 7. CO concentration along the vertical line, in correspondence of the measuring point.

CONCLUSIONS

The main characteristics of the flows (vortices and channelling) in the problem of CO diffusion within the real city of Rio de Janeiro are captured by both the RANS and the LES turbulence models, while small vortices are captured only by LES. The CO distributions obtained by both models are lower than the measured CO on ground level, but a longer sequence of measured data in a day is needed for a better comparison.

REFERENCES

Anand Y., Gupta, A., Tyagi,S.K. Anand,S. 2016: Computational fluid dynamics,a building simulation tool for achieving sustainable buildings: *Renewable and Sustainable Energy Reviews*, **57**, 1174-1185.

Di Sabatino, S., Buccolieri, R., Pulvirenti, B., Britter, R. E. 2008: Flow and Pollutant Dispersion in Street Canyons using FLUENT and ADMS-Urban: *Environ Model Assess*,**13,** 369-381.

Pimentel, L.C.G., Marton, E., Silva Soares, M., Jourdan, P. 2014: Characterization of surface wind regime in the Metropolitan Area of Rio de Janeiro. *Eng Sanit Ambient*, **19**, 121-132.

Santiago, J. L., Dejoan, A. , Martilli, A., Martin, F. Pinelli, A. 2010: Comparison Between Large-Eddy Simulation and Reynolds-Averaged Navier–Stokes Computations for the MUST Field Experiment. Part I: Study of the Flow for an Incident Wind Directed Perpendicularly to the Front Array of Containers. *Boundary-Layer Meteorol.* **135**, 109-132.