The influence of wind direction and source location in the flow pattern and concentration field over an urban canopy

Gabriel G. Almeida[1], Fernanda C. Cezana[2]

[1]*Dept. of Mechanical Engineering, Federal Institute of Espírito Santo*
[2]*Dept. of General Teaching, Federal Institute of Espírito Santo*
Highway BR 101 North – Km 58 – Litorâneo Neighbourhood, 29932-540, São Mateus/ES, Brazil
gabrielgusmaoalmeida@hotmail.com, fecezana@ifes.edu.br

Abstract. The flow characteristics and pollutant dispersion around a group of obstacles are investigated numerically using a Reynolds stress turbulence closure model. The mathematical modeling is based on the solution of the conservation equations (mass, momentum, and chemical species). Numerical simulations were performed over an idealized urban canopy with twenty-four unequally spaced buildings with rectangular cross sections and constant height with a commercial software ANSYS-FLUENT 16.0, that uses a finite volume method. The influence of the source location was evaluated. Two different locations of the point sources were considered. Wind direction influence was also analyzed and discussed. It was found that the flow pattern is very affected by the wind direction, with the formation of eddies and low-velocity regions, and the mean concentration distribution was also determined by the source location and building configuration relative to the wind.

Keywords: CFD, Urban canopy, Source location, Wind direction.

1 Introduction

The dispersion of pollutants in urban areas is a topic that has been extensively studied by researchers in different parts of the world (Carpentieri and Robins [1]; Boppana, Xie and Castro [3]; Goulart [5]). Urban areas are generally characterized by being densely populated, having a large vehicle emissions and large industries that contributes to increase the levels of air pollution. In addition, the urban areas are characterized by the presence of buildings that strongly interfere in the flow pattern and the dispersion of pollutants in their surroundings.

The effect of the geometry of buildings and their distribution or arrangement in the urban area is one of the dominant factors in the flow patterns and pollutant dispersion (Boppana, Xie and Castro [3]; Garbero, Salizzoni and Soulhac [4]). Urban areas can also be determined by buildings of different heights (Xie, Coceal and Castro [7]) and the arrangement of these buildings can be done in various ways, considering, for example, different wind directions. Garbero, Salizzoni and Soulhac [4] showed that in the same configuration with different orientations of the external flow, there are completely different dispersion patterns.

Atmospheric flow in urban areas is extremely complex (Belcher [2]), so it is important to use adequate methodology and tools to take that into account. Among the turbulence models available, it is well known that modelling with Large-Eddy Simulation (LES) is more accurate than using the Reynolds Averaged Navier-Stokes (RANS). However, the computational cost of LES is quite high. In a recent study, Ramponi *et al.* [6] provided a detailed review of the literature for CFD studies of outdoor ventilation for generic urban. The authors highlighted that LES has a high computational cost and that RANS, despite of its limitations, is a good alternative even for more complex urban configurations.

Therefore, the main aim of the present study consists of investigating numerically the flow characteristics and pollutant dispersion around a group of obstacles using a RANS methodology with a Reynolds-stress turbulence closure model for two different wind directions and two different sources point locations.

2 Methodology

In the present paper, the air flow and point source (see location in Figure 1) dispersion over an idealized urban environment, was investigated numerically using RANS. Figure 1 shows the computational domain, that was defined according to the wind tunnel experiment developed by Carpentieri and Robins [1], with all buildings with constant height, *i.e.*, $H_b = 102$ mm. The buildings blocks occupy an area of 230×350 mm² each and the Reynolds number based on the building height in the experimental conditions was $Re \approx 1{,}7 \cdot 10^4$. The reference wind measured at 1 m height was $U_{REF} = 2{,}5$ m s^{-1}. The origin of the coordinate system is in the center of the model, i.e., at the ground of the major intersection. More details of wind tunnel experiments are described in Carpentieri and Robins [1].

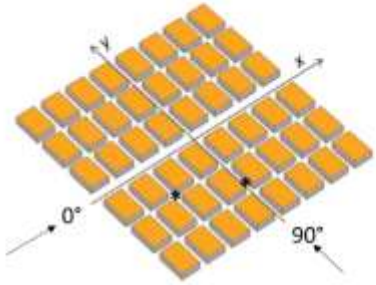


Figure 1. 3D representation of the model proposed by Carpentieri and Robins [1]. * represents source location.

2.1 Governing equations and numerical method

The governing equations for the atmospheric flow in neutral conditions considering an incompressible flow and a steady-state regime, based on the Reynolds Averaged Navier Stokes approach, are presented below.

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0, \tag{1}$$

$$\rho \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + g\delta_{i3} + \mu \frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j}, \tag{2}$$

where $\overline{u_i}$ is the mean velocity component in the i-th direction, ρ is the fluid density, μ is the absolute viscosity of the fluid, $\overline{p}$ is the mean pressure and g is the gravitational acceleration.

The transport equation for the scalar, considering its density the same as the air, is written as follows,

$$\frac{\partial \rho \overline{u}_i \overline{c}}{\partial x_i} = \frac{\partial}{\partial x_i}\left(D \frac{\partial \overline{c}}{\partial x_i}\right) + M, \tag{3}$$

where $\overline{c}$ is the mean concentration, D is the species total diffusivity and M is the source term. In the present study, the Schmidt number is set to 1. The non-dimensional mean concentration is calculated as: $\overline{c}^* = U_{REF} H_b^2 \overline{c}/Q$, where Q is the source flow rate.

To close the system of equations, the unknown term $\tau_{ij} = -\rho \overline{u_i' u_j'}$ should be modeled. In this work, we used the Reynolds-stress Omega model (Wilcox [8]), that is a Reynolds Stress Models (RMS) based on ω equation. Numerical simulations were carried out using the finite volume method present in the ANSYS Fluent 16.0.

2.2 Boundary conditions

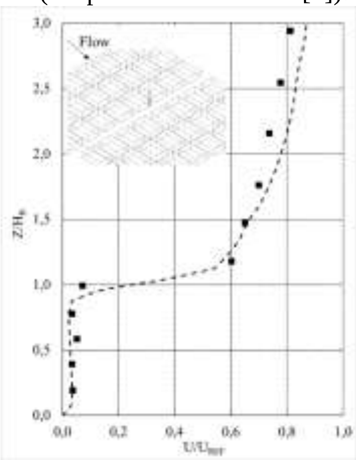
Velocity profile at inlet boundary was characterised by using the wind tunnel measurements as described in Carpentieri and Robins [1]. The turbulent kinetic energy profile (k) and specific dissipation rate profile (ω) were estimated at inlet by $k = u_*^2/\sqrt{C_\mu}$ and $\omega = \sqrt{k}/\sqrt[4]{C_\mu}\kappa z$, respectively, where u_* is the friction velocity, κ is the Von Karman constant, z is the height above the ground and C_μ is a constant. Mean concentration at inlet is $\overline{c} = 0$.

The pressure-outlet condition was used at the outlet. The flux for scalar is zero at the outlet. The top and laterals were treated as a symmetry plane. The boundary conditions for the buildings walls and bottom were no-slip. The scalar is set up as no flux at the buildings walls and bottom.

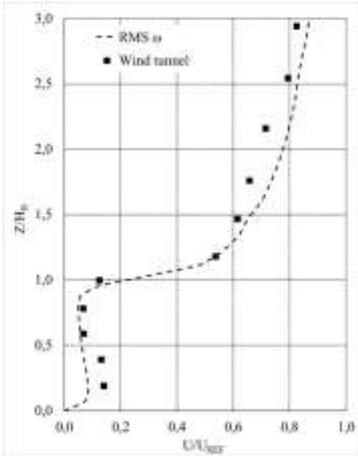
3 Results and discussions

3.1 Validation study

Vertical profiles of the mean horizontal velocity at two positions, obtained using the Reynolds-stress Omega Model (RMS ω), are shown in Figure 2. The results of the RMS ω are in satisfactory agreement with wind tunnel measurements (Carpentieri and Robins [1]).



(a)



(b)

Figure 2. Vertical profile of the mean horizontal velocity at $y/H_b = -0{,}88$, wind direction 90° at (a) $x/H_b = -2{,}30$ and (b) $x/H_b = -2{,}70$, for the present study (RMS ω) and wind tunnel (Carpentieri and Robins [1]).

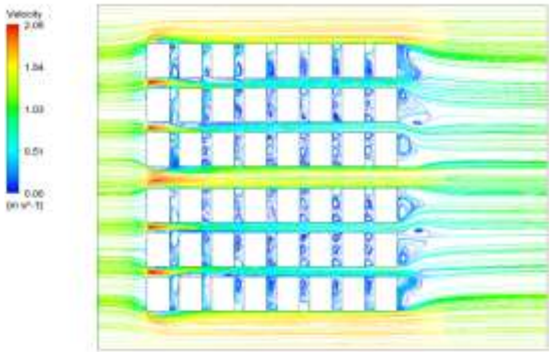
3.2 Characteristics of flow field: influence of wind direction

Figure 3 shows the streamlines of the mean velocity for different heights above the ground. Buildings can be identified as white. In Figure 3(a) the recirculation zones are evident. In Figure 3(b) the flow field is less influenced by the buildings since most streamlines only slightly change its directions. In the same way, Figure 3(c) shows formation of eddies between consecutive buildings, but in this case, the major street is now perpendicular to the wind direction which results in streamlines being redirected from one street to another, which could contribute to pollutant transport from one street to another. For the 0° case, wind speeds within the canopy appear to be generally higher. This result agrees with Carpentieri and Robins [1] wind tunnel measurements. According to the authors, the main reason for this result is probably that, in this case, the wind direction is aligned with the wider X street, where higher speeds are expected.

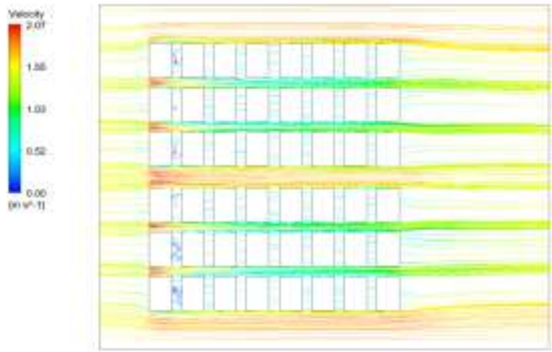
3.3 Characteristics of concentration field: influence of wind direction

Figure 4 presents non-dimensional mean concentration, c^*, at $z/H_b = 0{,}5$ and $z/H_b = 1$ for two different wind directions (0° and 90°). Figure 4(a) and (b) show mean concentration field for 0° case. Ground sources are located at one intersection and at between two buildings, as indicated in the Figure 4(a). Most pollutant from the source that is located at the intersection is channeled by the street while pollutant from the source that is located between two buildings is trapped by the recirculation zone in this region. In the last case, the pollutant that is trapped by the recirculation zone seems to be a kind of secondary source, as pointed out by Goulart [5]. According to the author, the scalar that enters the recirculation area will be trapped and released in a different time scale.

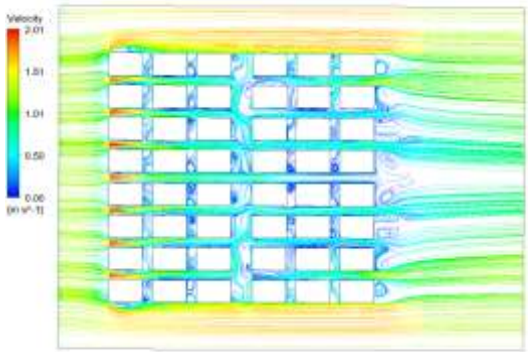
Figure 4(c) and (d) show mean concentration field for 90° case. In this case, ground sources are located at one intersection and in a main street, as indicated in the Figure 4(c). Pollutant from the source that is located on the main street behaves in the same way as the 0° case, i.e., scalar is channeled by the main street. On the other hand, pollutant from the source near an intersection is trapped by the recirculation zones in the vicinity. However, after the next side street, the scalar is channeled by a main street. Figure 4(d) shows the mean concentration of the scalar at the top of the buildings. In this case, the scalar disperses more than in the 0° case.



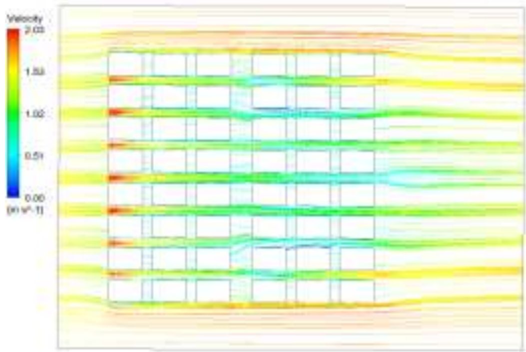
(a) 0°



(b) 0°



(c) 90°



(d) 90°

Figure 3. Plan view of the flow field streamlines at (a) and (c) $z/H_b = 0{,}5$; (b) and (d) $z/H_b = 1$.

4 Conclusions

The influence of wind direction was very much noticeable, given that the buildings have uneven dimensions, and the streets also have different widths. The presence of a wider street parallel to the wind contributes to the appearance of a flow short-circuit, reducing pollutant concentration when the source is in this area, moreover, the building's dimensions contribute to the number of eddies that appear downwind, such that with more eddies, harder it is to pollutant dispersion to occur, increasing the concentration in these regions. When the major street was put perpendicular to the flow, it redirected the flow from one street to a parallel one, which not only affected the flow field but also the concentration distribution coming from the source, that got redirected to a parallel street.

Wind direction, source location and building dimensions can strongly determine if pollutant will be carried away by the flow, causing little to no harm to people, or will be accumulated or even deflected to streets nearby, increasing the chances of the emergence of respiratory diseases. Through study of local wind direction and determination of the possible locations of pollutant sources, the arrangement of the buildings can be made such that the concentration throughout the region remains at an acceptable level, increasing the local air and health quality.

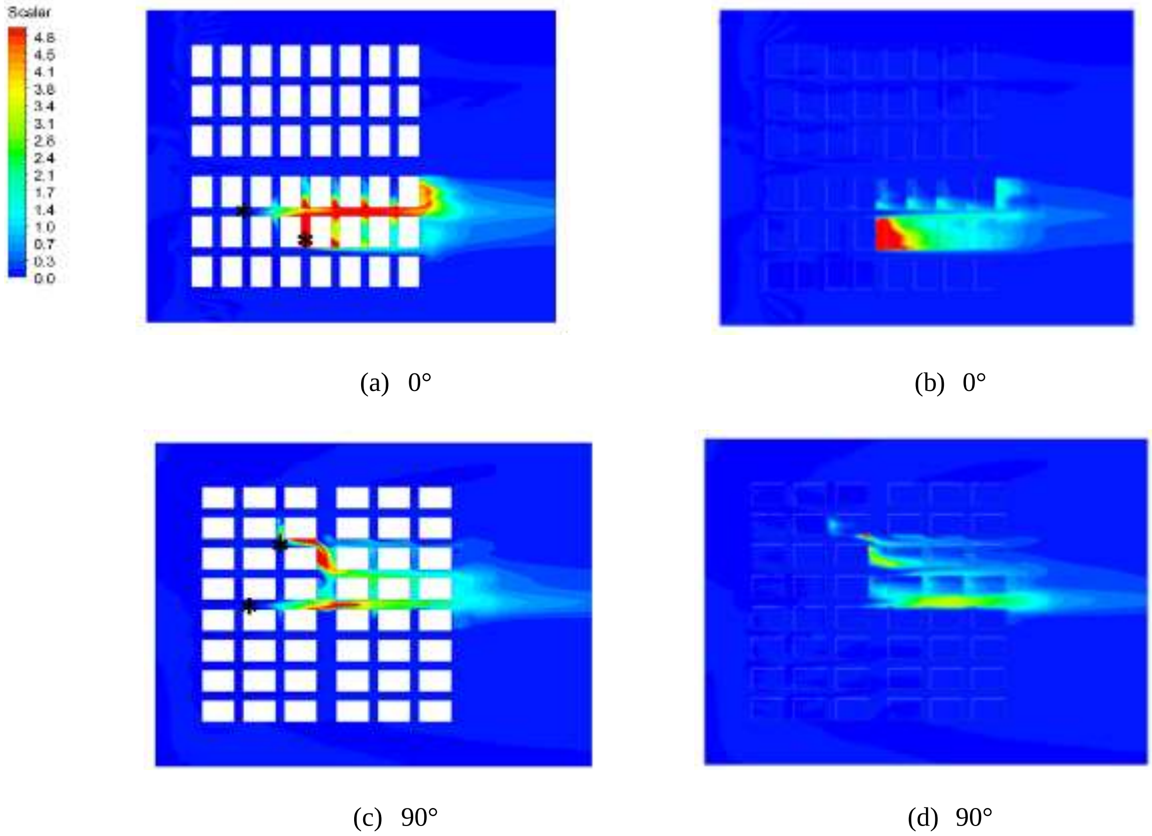


Figure 4. Plan view of the non-dimensional mean concentration c^* field at (a) - (c) $z/H_b = 0{,}5$ and (b) - (d) $z/H_b = 1$. * represents source location.

Acknowledgements. The authors gratefully acknowledge funding from Espírito Santo Research Foundation (FAPES) (Grant Term 080/2019) and Federal Institute of Espírito Santo (IFES).

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] Carpentieri, M., and Robins, A. G. “Influence of urban morphology on air flow over building arrays”. *Journal of Wind Engineering & Industrial Aerodynamics*. 145, 61–74, 2015.
[2] Belcher, S. “Mixing and transport in urban areas”. *Philosophical Transactions of the Royal Society A*, 363, 2947–2968, 2005.
[3] Boppana, V.B.L., Xie, ZT. & Castro, I.P. “Large-Eddy Simulation of Dispersion from Surface Sources in Arrays of Obstacles”. *Boundary-Layer Meteorol* 135, 433–454, 2010.
[4] Garbero, V., Salizzoni, P., Soulhac, L. “Experimental Study of Pollutant Dispersion Within a Network of Streets”. *Boundary-Layer Meteorol* 136, 457–487, 2010.
[5] Goulart E. V., Flow and dispersion over urban areas. 2012. 1-157 f. University of Reading, 2012.
[6] Ramponi, R., Blocken, B., de Coo, L. B., and Janssen, W. D. “CFD simulation of outdoor ventilation of generic urban configurations with different urban densities and equal and unequal street widths”. *Building and Environment*. 92, 152–166, 2015.
[7] Xie, ZT., Coceal, O. & Castro, I.P. “Large-Eddy Simulation of Flows over Random Urban-like Obstacles”. *Boundary-Layer Meteorol* 129, 1, 2008.
[8] Wilcox. D. C. Turbulence Modeling for CFD, Third Edition. DCW Industries, Inc., 2006. ISBN 978-1- 928729-08-2 (1-928729-08-8).