ENC-2022-0382
NUMERICAL STUDY OF DISPERSION OF CARBON MONOXIDE IN A STREET CANYON

Gabriel Gusmão Almeida
Tomaz Antônio Lisboa e Silva
Fernanda Capucho Cezana
Federal Institute of Science and Technology of Espírito Santo
gabrielgusmaoalmeida@hotmail.com
tmz.lisboa@hotmail.com
fecezana@ifes.edu.br

Abstract. The objective of this paper is to study the air flow and the carbon monoxide dispersion coming from vehicles within an idealized street canyon. The mathematical modelling used is based on the numerical solution of the mass, momentum and chemical species equations using the RANS methodology, which is used to model the turbulent flow. The finite volume method is applied through the commercial software Ansys Fluent 22.0. The results obtained were verified and compared with experimental results available in the specialized literature, which will determine the accuracy of the numerical model used and contribute to understand the carbon monoxide behavior inside an street canyon.

1. INTRODUCTION

As stated by Shi *et al.* (2020), the growth of great cities including buildings and skyscrapers brings us to the problem of transportation and that means many more vehicles, like cars and motorcycles that are responsible for many of the pollutants that we breathe.

According to the World Health Organization (2019) (WHO), 99% of the population of the world lived in places where the air quality guidelines levels were not met, besides that, the air pollution in both cities and rural areas has caused 4.2 millions premature deaths worldwide in 2016.

The height and arrangement of the street canyon are extremely relevant for helping or to disturb the dispersion of the emitted gases (Maia *et al.*, 2019), so, that means a deeper study of the variables that affect this phenomenon is important. Numerical simulation is present in this paper, as it is one of the most advanced and effective ways for solving mathematical problems nowadays, with the objective of making better planning of changes that can improve better breath health of people inside the canyon.

Many pollutant compose the inventory of vehicular emission as Nitrogen Oxides (NOx), Sulfur Dioxide (SO_2), Carbon Monoxide (CO), Volatile Organic Compounds (VOCs), greenhouse gases and suspended particles. However, CO is the most produced by vehiclesTobón *et al.* (2020), and that is why it was chosen for this paper. Furthermore, Taseiko *et al.* (2009) pointed out that carbon monoxide gas is a good indicator of the dispersion and dilution of pollutants from vehicles, since its chemical response time is quite long.

The problem of pollutant dispersion of traffic carbon emissions can be studied in three ways: field measurements, wind tunnel experiments or numerical simulations. To collect data (field measurements or wind tunnel), high tech instruments with a significant precision are required, which makes this experiment with elevated cost and almost impracticable at some cases. (Shi *et al.*, 2020).

The objective of this paper is to analyse numerically the flow and dispersion of carbon monoxide from standing vehicles in an idealized street canyon using a Reynolds Averaged Navier-Stokes (RANS) methodology.

2. METHODOLOGY

In the present paper, the air flow and dispersion over an urban street canyon was investigated numerically using RANS. Fig. 6 shows the computational domain that was defined according to the wind tunnel experiment developed by Kastner-Klein *et al.* (2001). Two building rows were mounted in the neutrally stratified wind tunnel, simulating an isolated street

canyon with aspect ratio, H/W = 1, and the length-to-depth, L/H = 10 (H: height, L: length of the buildings and W: street width). The height of the buildings forming the canyon has been 12 cm, and their length has been 120 cm. The configuration considered in this paper is an external wind flow that has been directed perpendicular to the axis of the canyon, see Figure 1, and the X axis has been oriented along the direction of external wind.

This paper considered a twelve standing vehicle configuration at street level, within the street canyon, but only the external wind was considered. The vehicles were modeled as rectangular obstacles having dimensions $l_x = 10\,\text{mm}$, $l_y = 3\,\text{mm}$ e $h_v = 8\,\text{mm}$, according to the wind tunnel configuration of Kastner-Klein *et al.* (2001). Point sources were located behind each vehicle at half the vehicle height in order to investigate the pollutant dispersion within the street canyon. More details of wind tunnel experiments are described in Kastner-Klein *et al.* (2001).
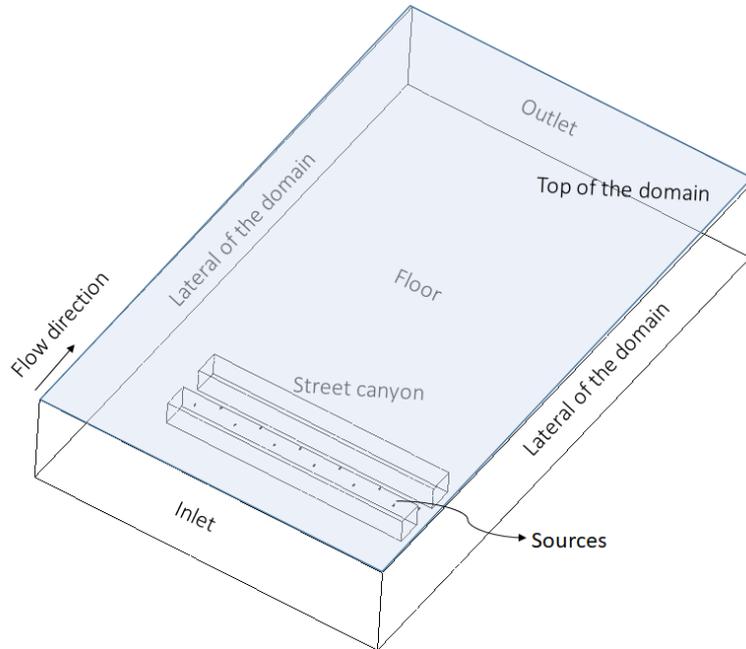

Figure 1. 3D representation of the model according to Solazzo *et al.* (2008)

2.1 Mathematical model

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 (RANS) approach were used and are presented below.

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

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

where $\overline{u_i}$ is the mean velocity component in the i 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. τ_{ij} is known as the Reynolds Stress Tensor.

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 0.7.

The turbulence closure model used was the Standard $k - \epsilon$, (Launder and Spalding, 1996) and the constants used in the model are: $C_\mu = 0.09$, $C_{1\epsilon} = 1.44$, $C_{2\epsilon} = 1.92$, $\sigma_k = 0.53$, $\sigma_\epsilon = 0.55$.

2.2 Boundary conditions

The boundary conditions used in the simulations are displayed in the Table 1, where $\kappa = 0.40$ is the von-Karman constant, $C_\mu = 0.09$ is a model parameter, and $U(z)$ the velocity at height z above the ground. Values of friction velocity $u_* = 0.43\,\text{m/s}$, roughness length $z_0 = 0.0007\,\text{m}$ and boundary layer depth $\delta = 0.48$ m were obtained from Kastner-Klein *et al.* (2001). The resulting free-stream velocity for the given profile is $u_0 = 7\text{m/s}$.

Table 1. Boundary Conditions

Inlet	$U(z) = \dfrac{u_*}{\kappa} \ln\left(\dfrac{z}{z_0} + 1\right);\ \ \epsilon = \dfrac{u_*^3}{\kappa(z + z_0)}\left(1 - \dfrac{z}{\delta}\right);\ \ k = \dfrac{u_*^2}{\sqrt{C_\mu}}\left(1 - \dfrac{z}{\delta}\right);$ Zero flux for scalar
Outlet	outflow
Buildings, floor and vehicles	no-slip; no flux for scalar
Laterals and top	free-slip; no flux for scalar

2.3 Mesh and computational procedure

Zheng-Tong and Castro (2008) conducted a study about different mesh types, as tetrahedral mesh, hexahedral mesh and polyhedral mesh. Their results showed that polyhedral mesh in comparison with hexahedral mesh are essentially identical. However, the number of cells used in the former was much smaller. Furthermore, the results obtained by Zheng-Tong and Castro (2008) confirm that the polyhedral mesh is more accurate and less memory consuming than the widely used tetrahedral mesh. In this work, a polyhedral mesh (2.5 million cells) was implemented for the numerical simulation, using the Fluent Meshing 22.0 (see Figure 2).
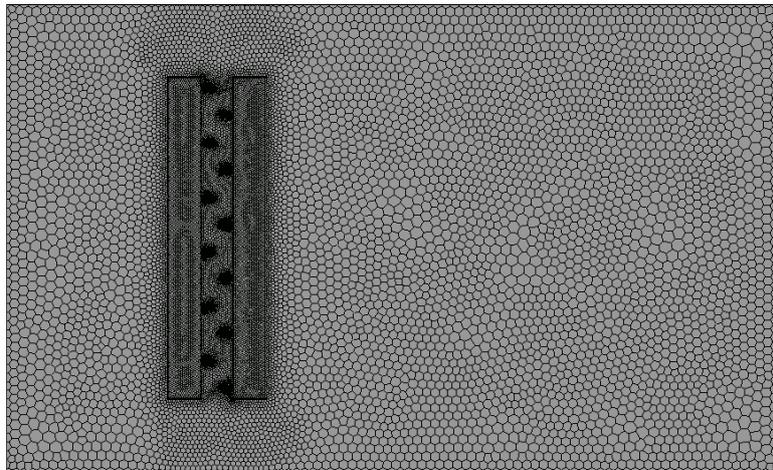

Figure 2. Plane view, in the x - y plane, of the computational mesh.

Figure 3 shows a detailed view of mesh refinement on the vehicles and sources.

The equations are numerically solved in steady-state regime, using the finite volume method of Ansys Fluent 22.0 computational code. The pressure-velocity coupling used was the Coupled scheme, since it resulted in better residual stability. The spatial discretization setting up was second order upwind for all variables. The relaxation coefficients were set as default by the Ansys 22.0.

3. RESULTS AND DISCUSSION

3.1 Model validation

To evaluate the numerical data obtained in this study, comparisons with wind tunnel data from Kastner-Klein *et al.* (2001) were done. Data were collected at the vertical plane $y/H = L/2$, for two horizontal positions throughout the canyon width, corresponding to x/H = 0.000 (the middle of the canyon) and x/H = 0.375 (windward position).

Figures 4 e 5 show the comparison between the mean horizontal and vertical velocity profiles simulated numerically and the wind tunnel measurements. The comparison of the modelled mean horizontal velocity with wind tunnel data was overall satisfactory (see Figure 4). However, close to street level the model performed not very well. On the other hand,

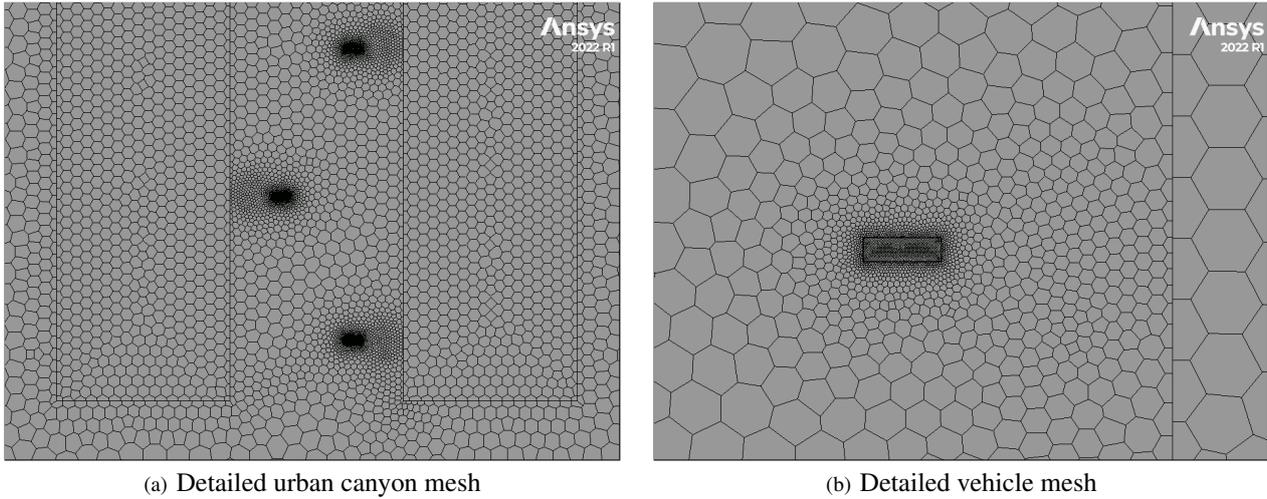(a) Detailed urban canyon mesh

(b) Detailed vehicle mesh

Figure 3. Detailed mesh

the model is very satisfactory for $Z/H \leq 0.5$ in the case of mean vertical velocity.

Figure 6 shows a comparison between the numerical data obtained in the present work, wind tunnel data from Kastner-Klein *et al.* (2001) and a numerical simulation data (using RANS) obtained by Solazzo *et al.* (2008). The peaks of the turbulent kinetic energy (TKE) at the roof level ($z/H \approx 1$) were reasonably reproduced by both numerical models. As pointed out by Solazzo *et al.* (2008), this indicates that the model can reliably simulate the flow region where shear stresses are dominant, such as the shear layer on the top of the canyon. In that region, the TKE decreases from the middle of the domain (6(a)) to the windward side (6(b)). However, inside the canyon the TKE magnitude is underestimated by both numerical models. That indicates the model needs to be improved.

Overall, the results showed a reasonable agreement with wind tunnel and the numerical simulation by Solazzo *et al.* (2008).
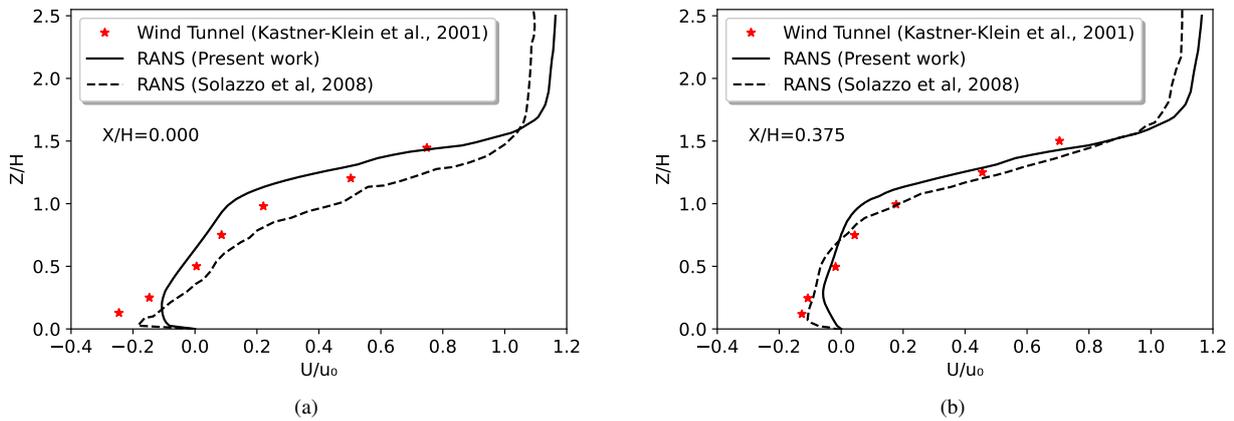

Figure 4. Comparison between modelled and measured mean horizontal velocity.

3.2 Flow and dispersion within the street canyon

Figure 7 shows the velocity vectors in the x - z plane located at $Y/H = 0$. In this study the aspect ratio was H/W = 1. The flow is characterised by the formation of a single vortex within the canyon in which case the flow within the canyon becomes decoupled from the flow above the buildings. This phenomena is called skimming flow Oke (1988) and it was expected, given the aspect ratio based of the building's length and height.

Figure 7 depicts the perpendicular flow approaching the lined up pair of buildings, where two different regions appear inside the canyon. The first region is the windward side with downdraft motion responsible for carrying clean air from above the building, and the second region is the leeward side characterized by updraft flow in the other side, which is responsible to transport polluted air from inside the canyon to the outer zone.
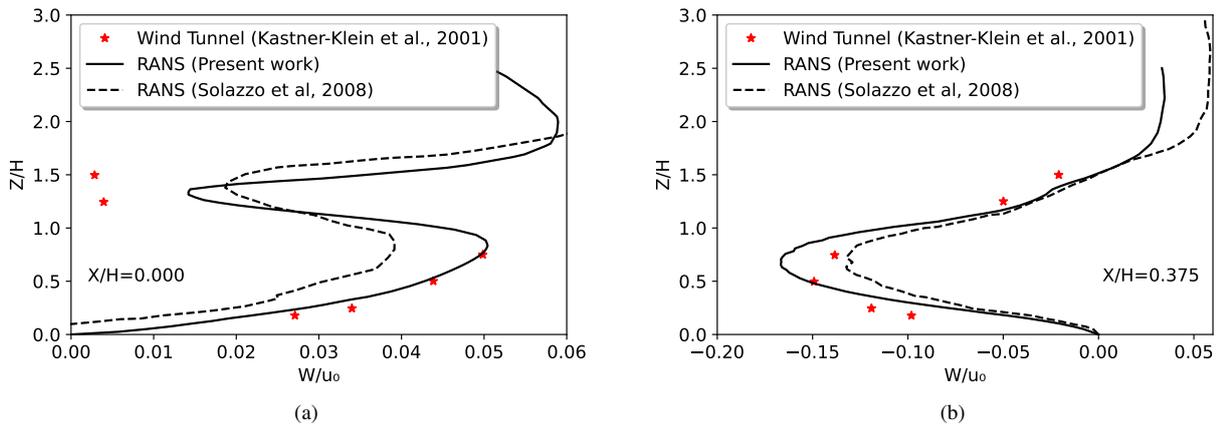
(a)

(b)

Figure 5. Comparison between modelled and measured mean vertical velocity.
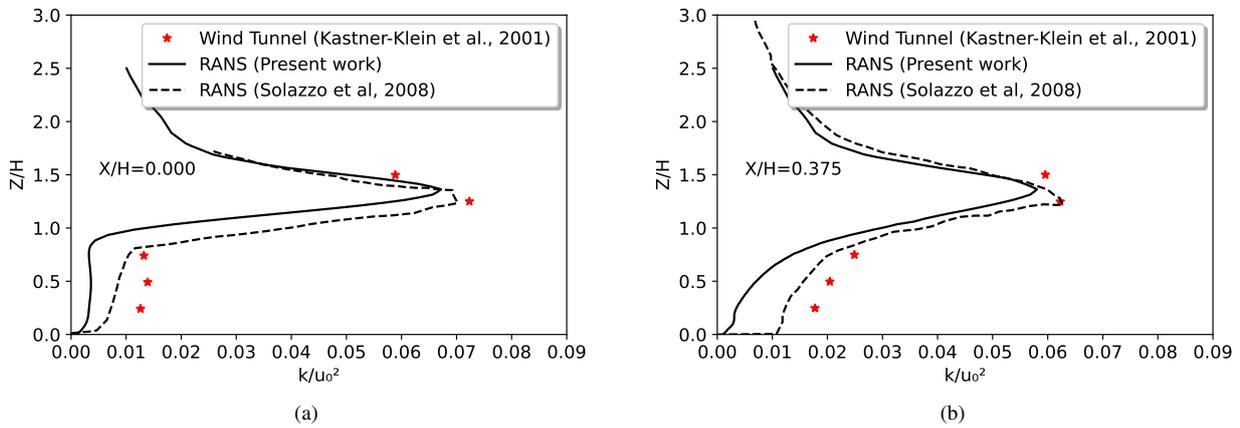

(a)

(b)

Figure 6. Comparison between modelled and measured mean TKE profiles.
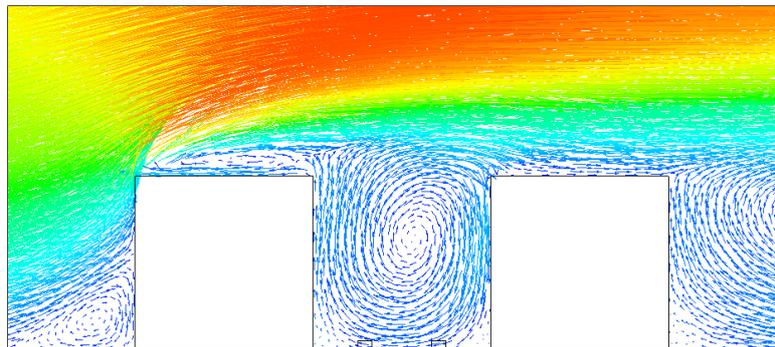

Figure 7. Velocity vectors in the x - z plane located at $Y/H = 0$

Figure 8 shows a contour of mean concentration of monoxide carbon (CO) within the street canyon in the x - z plane at $Y/H = 0$. The results obtained in this study are similar to those acquired in the experiments by Kastner-Klein *et al.* (2001), showing quite uniform values within the canopy, except close to the source and the walls.

In the Figure 8 it is possible to see that higher concentration are detected close to the upwind all whereas lower concentration are detected close to the downwind wall. This is coherent with the flow velocities, since in the downwind wall there is a inflow of fresh air from the external flow that dilutes the pollutant.
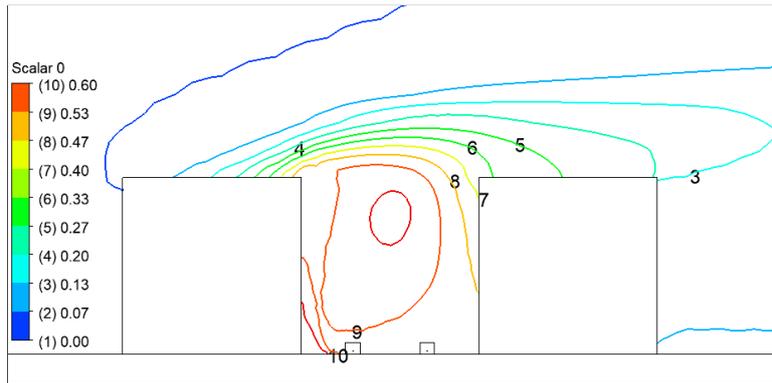
Figure 8. CO contour in the x - z plane located at $Y/H = 0$.

Figure 9 shows a isosurfaces of CO concentration in g/kg. The plume is carried by the flow in its main direction (left to right). The results shown in this figure are in accordance with the discussion about Figure 8. However, the plume of pollutant reaches first the downwind building (9(a)) and after goes to the top of the upwind building (9(d)), where the plume is more diluted.
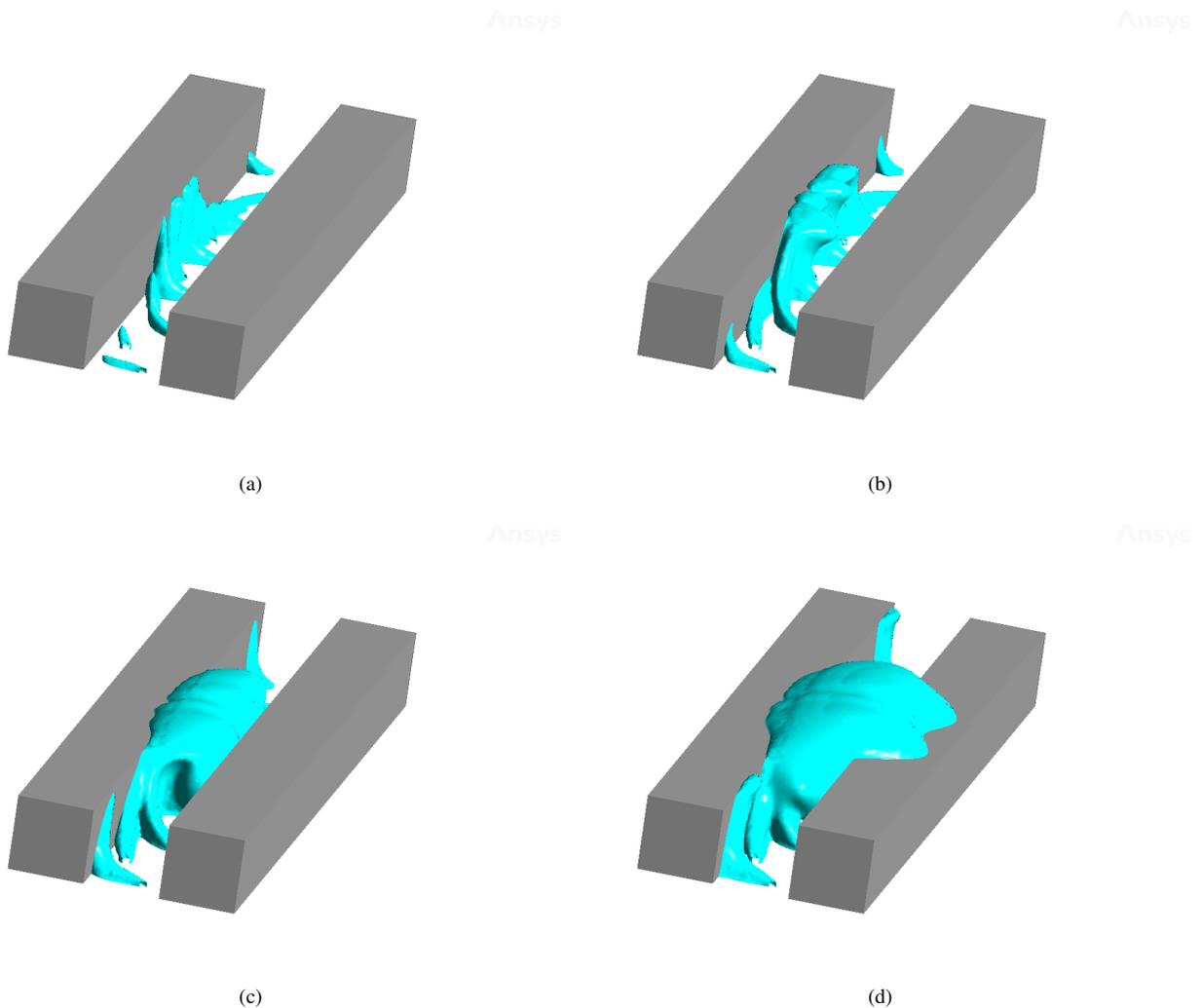

(a)

(b)

(c)

(d)

Figure 9. Isosurface of CO concentration (a) 0.5 g/kg, (b) 0.25 g/kg, (c) 0.125 g/kg and (d) 0.0625 g/kg.

4. CONCLUSIONS

This paper showed that the Standard $k - \epsilon$ model described well the turbulent fluid flow, mainly the vertical velocity component inside the street canyon, given its low computational cost compared to others RANS models. In addition, the

carbon monoxide dispersion showed that it is greatly affected by the flow since the advective transport dominates over other effects such as gravity and molecular diffusion, given that carbon monoxide and air have about the same density and the turbulent diffusivity generated, associated with the induced external flow is the main responsible for carrying out the pollutant.

In future studies the authors aim to analyze the influence of other variables such as the vehicle speed, in order to model the real influence of the vehicle in the dispersion of the pollutant within the canopy and how this parameter affect human health.

5. ACKNOWLEDGEMENTS

6. REFERENCES

Berkowicz, R., Hertel, O., Larsen, S., Sørensen, N.N. and Nielsen, M., 1997. "Modelling traffic pollution in streets". *National Environmental Research Institute, Roskilde, Denmark*, Vol. 10129, No. 10136, p. 20.

Cezana, F.C., 2015. *Estudo da transferência de massa de poluente no topo de um dossel urbano*. Ph.D. thesis, Universidade Federal do Espírito Santo, Vitória, Brasil.

Companhia Ambiental do Estado de São Paulo - CETESB, 2019. *Relatório Emissões veiculares no estado de São Paulo*.

Conselho Nacional do Meio Ambiente - CONAMA, 2018. *Resolução nº 491 de 19 de novembro de 2018. Dispõe sobre padrões de qualidade do ar. Diário oficial da República Federativa do Brasil*. Brasília.

Ganguly, R. and Broderick, B.M., 2010. "Estimation of co concentrations for an urban street canyon in ireland". *Air Qual Atmos Health*, Vol. 3, pp. 195–202.

Kastner-Klein, P., Fedorovich, E. and Rotach, M.W., 2001. "A wind tunnel study of organised and turbulent air motions in urban street canyons". *Brasileira de Gestão Urban*.

Machado, V., 2019. *Estudo Numérico da dispersão de poluente veicular em uma microrregião urbana*. Master's thesis, Universidade Tecnológica Federal do Paraná, Ponta Grossa.

Maia, J.L., Netto, V.M. and COSTA, B.L.G., 2019. "Forma urbana e poluição atmosférica: Impact os na cidade do rio de janeiro". *Brasileira de Gestão Urban*, Vol. 11. doi:https://doi.org/10.1590/2175. 3369.011.e20180145.

Nicholson, S.E., 1975. "A pollution model for street-level air". *Atmospheric environment*, Vol. 9, No. 1, pp. 19 – 31.

Oke, T.R., 1988. "Street design and urban canopy layer climate". *Energy and Buildings*, pp. 103 – 112.

Ramoul, L.B., Korichi, A., Popa, C., Zaidi, H. and Polidori, G., 2019. "Numerical study of flow characteristics and pollutant dispersion using three rans turbulence closure models". *Environ Fluid Mech*, Vol. 19, p. 379–400.

Shi, T., Sun, D., Zhang, Y., Xiong, J. and Zhao, Z., 2020. "Modeling emission flow pattern of a single cruising vehicle on urban streets with cfd simulation and wind tunnel validation". *International Journal of Environmental Research and Public Health*, Vol. 17, No. 12, pp. 4557–4574.

Solazzo, E., Cai, X. and Vardoulakis, S., 2008. "Modelling wind flow and vehicle-induced turbulence in urban streets". *Atmospheric Environment*, pp. 4919 – 4931.

Taseiko, O.V., Mikhailuta, S.V., Pitt, A., Lezhenin, A.A. and Zakharov, Y.V., 2009. "Air pollution dispersion within urban street canyons". *Atmospheric Environment*, Vol. 43, p. 245–252.

Tobón, A.M., Moncho-Esteve, I.J., Martínez-Corral, J. and Palau-Salvador, G., 2020. "Dispersion of co using computational fluid dynamics in a real urban canyon in the city center of valencia (spain)". *Atmosphere*, Vol. 11, No. 7, pp. 693–710.

Vardoulakis, S., Fisher, B.E.A., Pericleos, K. and Gonzales-Flesca, N., 2003. "Modelling air quality in street canyons: a rewiew". *Atmospheric Environment*, Vol. 37, pp. 155–182.

World Health Organization, 2019. "Ambient (outdoor) air pollution". Disponível em: `https://www.who.int/news-room/fact-sheets/detail/ambient-(outdoor)-air-quality-and-health`. Acesso em: 22 fev. 2022.

Zheng-Tong, X. and Castro, I.P., 2008. "Les for street-scale environments and its prospects". *Quality and Reliability of Large-Eddy Simulations*, Vol. 12.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.