

PAPER • OPEN ACCESS

Modelling of distribution of aerosol emission with account of buildings and structures of NPPs

To cite this article: M Mehdi *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **487** 012011

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

Modelling of distribution of aerosol emission with account of buildings and structures of NPPs

M Mehdi^{1,2}, M P Panin¹ and B Mohammedi²

¹National Research Nuclear University MEPhI (Moscow Engineering Physics Institute), Moscow, Russia

²Nuclear Research Center of Birine, Algeria

E-mail : mehdiyahia1@hotmail.fr

Abstract. In the lack of experimental data and due to the rapid development of numerical modelling and computer hardware, computational fluid dynamics (CFD) is widely used to study wind field and contaminant transport around buildings. The ability to predict quantitative values for these processes is not so clear. CFD simulations work under neutral atmospheric conditions and are validated by the Gauss model. The Gaussian model is a mathematical distribution of the concentration of contaminants from NPP ventilation pipes. It is the most recommended model by the IAEA and most widely used because 1) it gives results consistent with experimental data, 2) it is quite easy to use, and 3) it is consistent with the aleatory nature of turbulence. Reynolds-averaged Navier-Stokes equations is the most widely used method in turbulent flow modeling and simulation. In this paper, the prediction of flow accuracy and propagation around buildings with a stack was examined using k - ϵ model. The numerical results were compared to Gauss model results. Contaminant dispersion was well predicted with k - ϵ model. It was also confirmed that the concentrations predicted by the CFD models were more diffused than those of the Gauss model and that the results depend on the position of the obstacles.

1. Introduction

The transport of atmospheric pollutants in the environment is influenced by many complex factors. Among them, are wind conditions and urban morphology which are the most relevant to pollutant dispersion. Examples of wind conditions are wind speed and turbulence intensity. The higher the wind speed, the more mixing of pollutants with fresh air and the greater is the dilution - in other words, the lower is the concentration of pollutants detected in the wind flow. As for urban morphology, the more complex it is, the more it increases vortex structures in the wake of buildings. These recirculation zones tend to trap pollutants and increase local concentrations, which presents a hazard when the fresh air intakes of a building are located in a contaminated area, as soon as the risk of ingesting pollutants is increased [1–3].

This episodic phenomenon is present in all high-density urban areas; however, the literature offers little information and recommendations to avoid the problems that arise from it. In order to limit indoor air contamination by ingestion of pollutants from the outside, a better understanding of pollution aerodynamics and mechanisms of transport is necessary.

The research methodology used employs computational fluid dynamics (CFD) to study the dispersion of pollutants around a group of buildings. This tool provides detailed information on flow profiles and concentration (or dilution) fields by solving the flow equations throughout the calculation range. The reliability of numerical simulations is one of the main concerns of this study. CFD is based on the analysis of fluid flows, mass transfer and related phenomena, solving a subset of Navier Stokes' conventional equations at finite locations on a grid. It simultaneously produces results relating to flow characteristics at all points in space. Based on fundamental physics, CFD is used in a wide range of basic research and practical applications. In the field of urban wind studies, CFD has proven to be a



promising technology because of its flexibility in modeling complex geometries, such as those of cities with high building densities. CFD is not inherently limited by similarity constraints (as in wind tunnels), so it should be possible to digitally simulate all aspects of pollutant dispersion and its interactions with the environment [4]. That said, while CFD offers some advantages, it requires special care to provide reliable results. A number of parameters, such as grid size, discretization scheme, choice of turbulence model and boundary conditions, need to be controlled and validated by systematic comparisons with experimental data or other highly accurate methods [5].

One of the first studies on the complexity of the flow field around a body (representing a single building) and the relative effectiveness of different models of turbulence was carried out by Murakami and Mochida [6]. In this study, the distribution of velocities obtained from three-dimensional simulations of continuous flow around a cubic model was compared with wind tunnel results to evaluate the accuracy of the standard $k-\varepsilon$ turbulence model. An examination of the distribution of kinetic energy (k) revealed that the production level of k at the windward angle of the model was largely overestimated. Among other subsequent research, the study suggested changes to turbulence modeling in the $k-\varepsilon$ model.

Meroney et al [7] studied the flow field and dispersion around several shapes of buildings. They compared the results obtained in the wind tunnel to the measurements obtained in the wind tunnel. The purpose of these comparisons was to determine whether commercial software could be used to correctly simulate engineering wind problems. The authors found that numerical simulations invariably overestimated the surface concentrations downwind of the emission source. They attributed the observed differences to the inability of the Reynolds numerical model, on average, to reproduce the intermittency of the flow in the recirculation zones visualized in the wind tunnel. Moreover, although the concentration profiles were well reproduced, the amplitudes were frequently higher by an order of magnitude than those measured in the wind tunnel. With regard to the profiles of pressure, numerical predictions have been shown to be reasonably accurate, and amplitudes sufficiently satisfactory to allow engineering calculations, implying that mean pressure fields are less sensitive than other criteria to features of the digital model. The study finally revealed that the turbulence model RSM produced somewhat more realistic results than the SKE and RNG models.

2. Gaussian Model

Is one of the most widely used models to represent numerically the movement and dispersion of effluent from an emission point. Figure 1 illustrates in a simplified way a Gaussian model representing the contaminants emitted by a pipe.

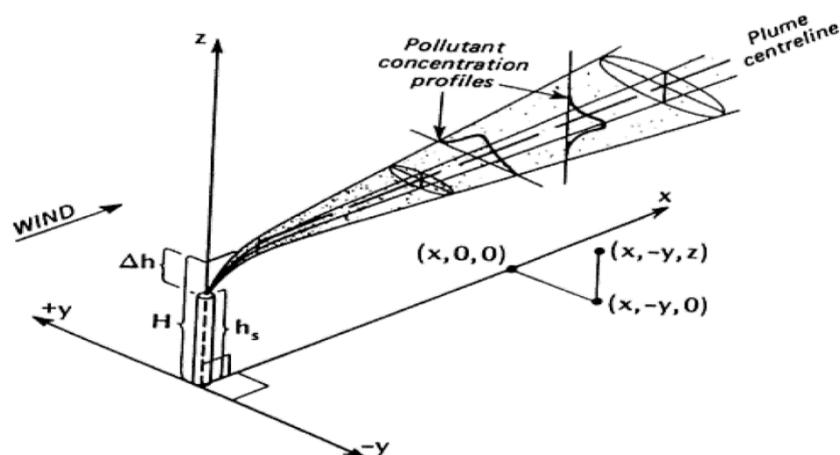


Figure 1. Plume dispersion coordinate system.

Gaussian dispersion model

$$C(x, y, z; H) = \frac{Q}{2\pi\sigma_y\sigma_z u} * \exp\left[-\frac{1}{2}\left(\frac{y}{\sigma_y}\right)^2\right] * \left[\exp\left[-\frac{1}{2}\left(\frac{z-H}{\sigma_z}\right)^2\right] + \exp\left[-\frac{1}{2}\left(\frac{z+H}{\sigma_z}\right)^2\right] \right]$$

where $C(x, y, z)$: average contaminant concentration at one point in the space in Bq/m^3 ;

Q : radioactivity flow rate emitted by the source in Bq/s ;

u : average horizontal wind speed in m/s at the height of emission;

σ_y et σ_z standard deviations of the horizontal and vertical Gaussian distribution in meters;

H : total plume elevation in meters which is equal to $h + \Delta h$: actual pipe height plus the immediate plume height.

The difficulty in a Gaussian model is the evaluation of turbulent diffusion. These are the standard deviations of distribution σ which represent the turbulent diffusion along the y and z directions in this equation. These coefficients depend on flow characteristics, turbulence, atmospheric stability and surface roughness. They are experimentally determined from large-scale trials. There are several correlations for determining standard deviations including those of Briggs [8], which depend on distance to source and atmospheric stability classes and those of Doury [9], which depend on transfer time.

3. CFD simulations

ANSYS-FLUENT 14.5 [10], has being used for simulation. The purpose of numerical predictions is to solve all partial differential equations applicable to the flow of any fluid, including the wind flow in the atmosphere. These equations are based on the fundamental laws of mass conservation and amount of motion (Navier-Stokes equations) in the computational domain.

3.1 The Computational domain

In this study the atmospheric dispersion is modeled around the Rostov nuclear station. The Rostov nuclear station consists of four blocks, the ventilation pipe is placed in the middle of the Y axis and 320 m from the X axis with a height of 100 m. The dispersion simulation area is 1520 m x 800 m x 300 m. For the dimensions and definition of the location of buildings and structures, we used satellite maps obtained from the Yandex site.

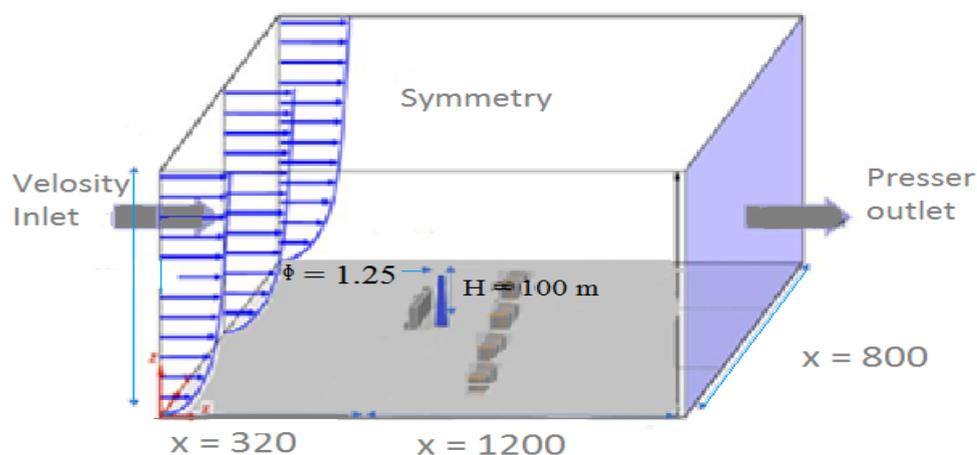


Figure 2. Schematic sketch of geometry and boundary conditions.

3.2 The Boundary Conditions

For the k-ε model, Durbin and Petterson Reif [11], proposed vertical profiles for mean wind speed *u*, turbulent kinetic energy *k* and turbulence dissipation rate ε.

$$u(z) = \frac{u^*}{K} \left[\ln \left(\frac{z}{z_0} \right) \right]$$

$$k = \frac{u^{*2}}{\sqrt{C_\mu}}$$

$$\varepsilon = \frac{u^{*3}}{Kz},$$

where *z* is the height co-ordinate, *u** the friction velocity, *K* the von Karman constant (0.40-0.42) and *C_μ* a model constant of the standard *k-ε* model.

4. Results and discussion

At the entrance of the simulation domain, the wind speed profile, in red on figure 3, gives a minimum speed of 0 m/s on the ground at *z* = 0 m and a maximum speed of 5.7 m/s at *z* = 100 m (the height of the stack). For the position between the blocks, at *x* = 270 m, the speed profile from *z* = 0 m to *z* = 60 m (the height of the buildings) the profile becomes complex due to the eddies created by buildings. Far away from buildings (*x* = 1200 m) the wind speed profile returns to ordinary shape.

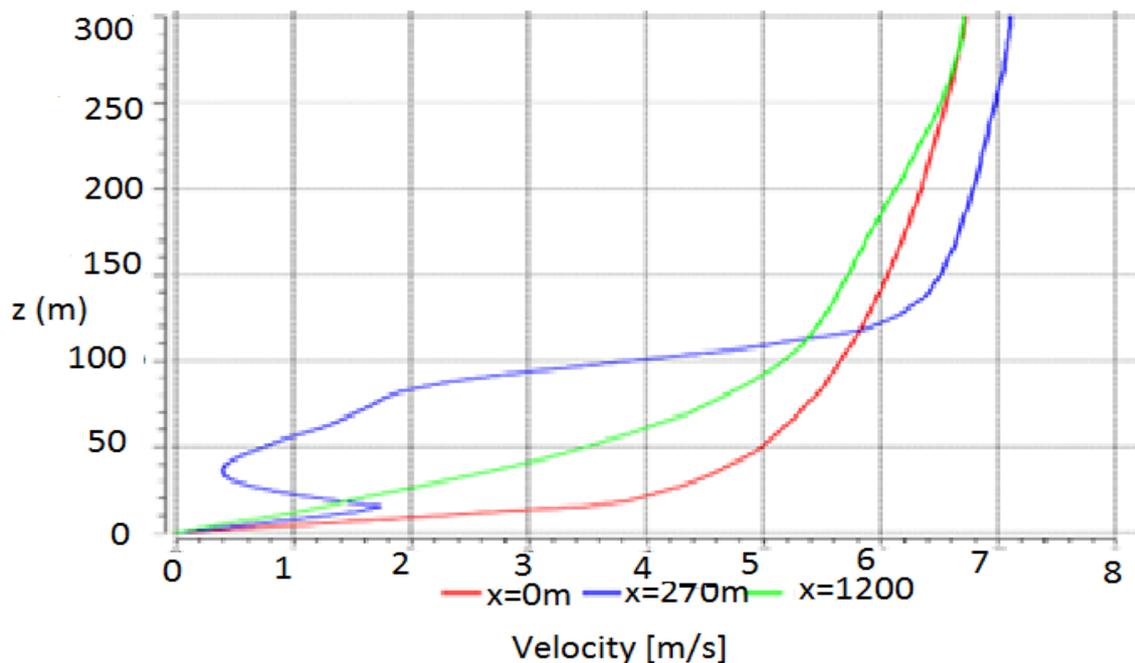


Figure 3. Simulated horizontal velocities at three locations.

A large recirculating vortex can be seen over the entire height of the stacks and blocks, at low speed (figure 4).

As expected and quite naturally the blocks influence the wind path around the site (figure 5). Buildings in their wake create recirculations (eddies) slowing the flow. The blocks downstream of the flow have an influence that can be noticed by the blue vectors (low speeds).

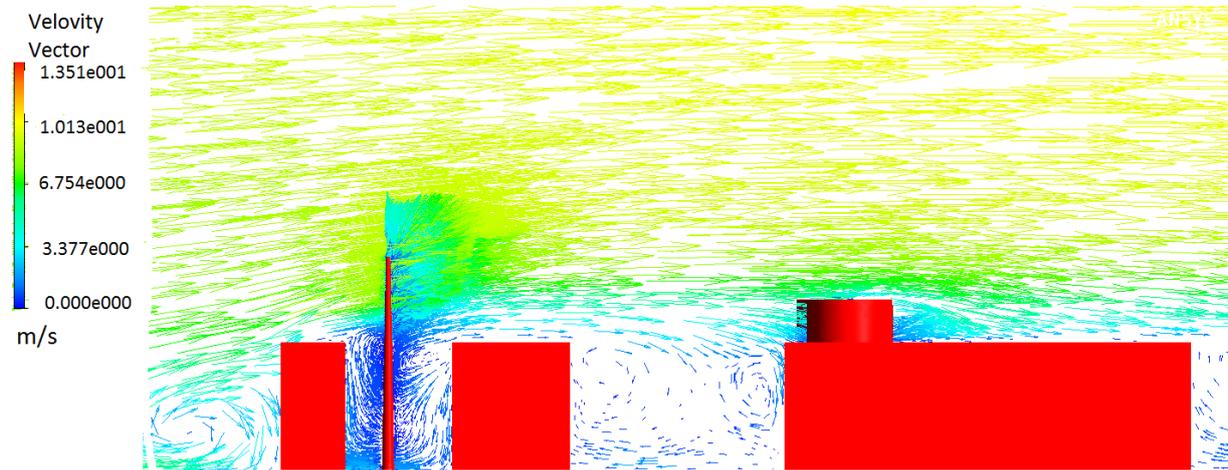


Figure 4. Velocity vectors (vertical cut).

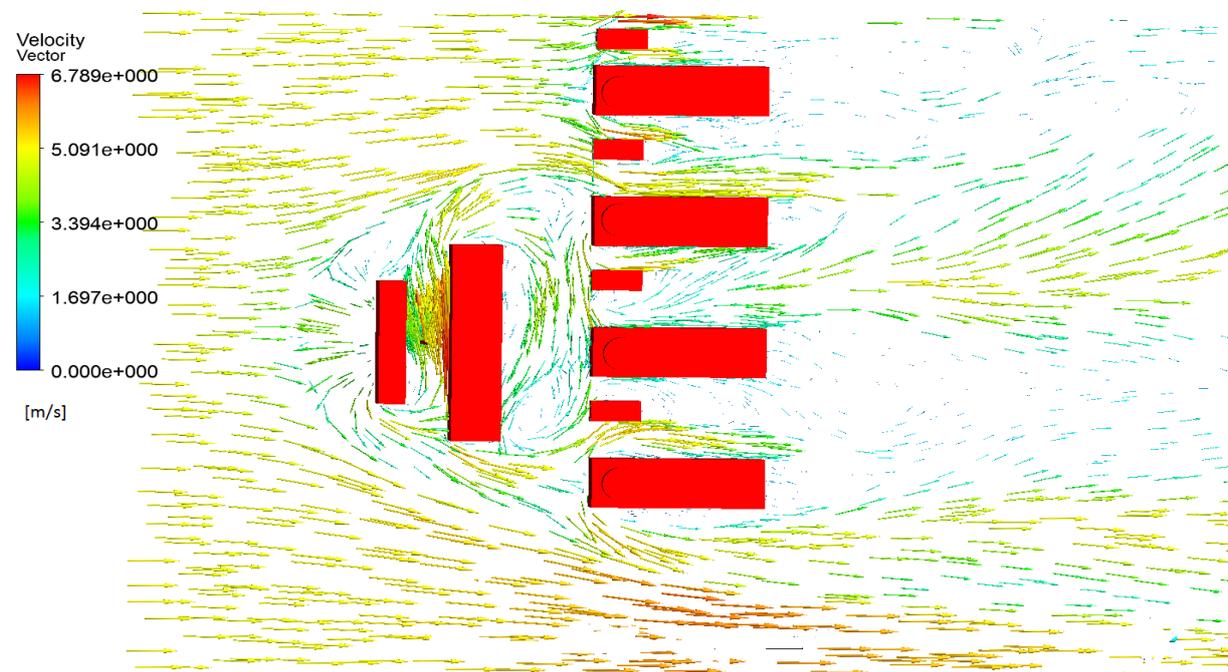


Figure 5. Velocity vector (top view).

Reduced interval between blocks creates a flow vortex which partially drives the flows (figure 6), contributing to an increase in their dilution downstream.

The blocks could have a "positive" effect on contaminant plume dilution. Indeed, the plume expanded vertically under the effect of the zone of turbulence created by blocks (figure 7). The plume is in a way less concentrated because it is more vertically extended.

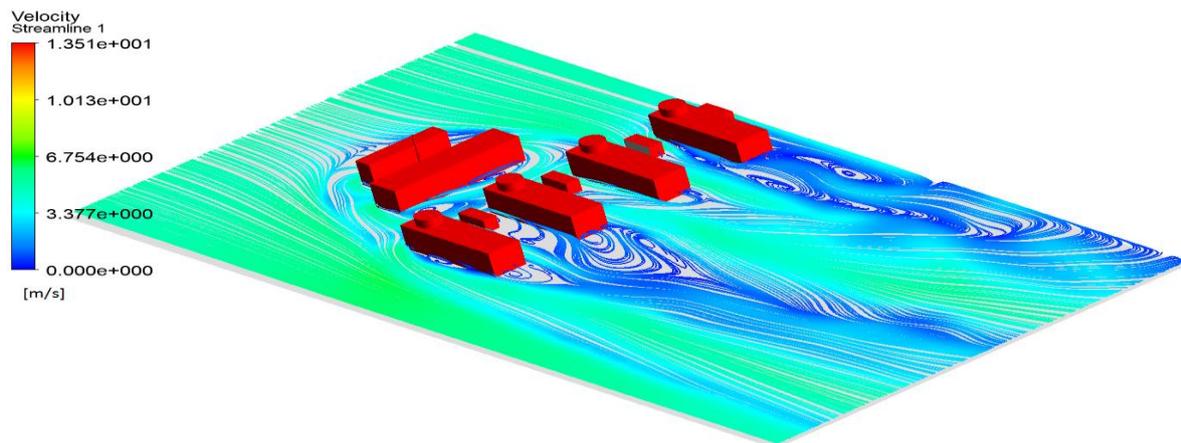


Figure 6. Streamlines velocity.

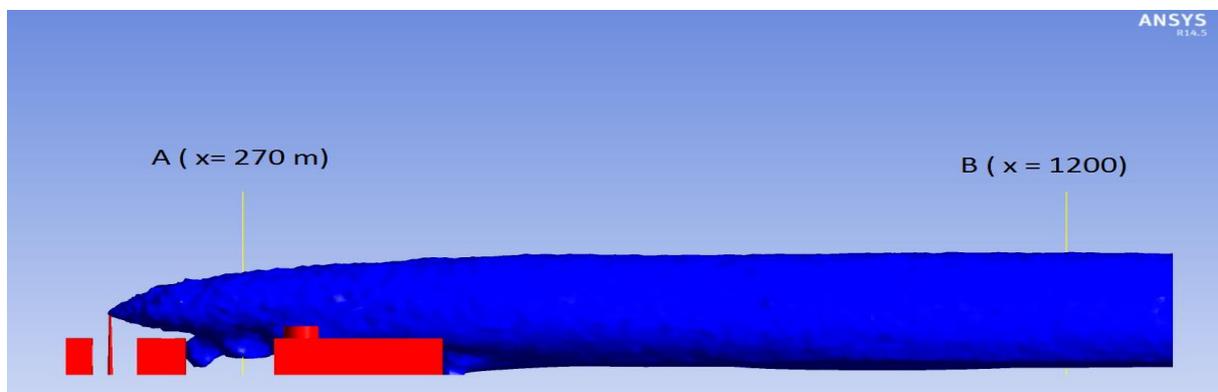


Figure 7. Isosurface of contaminant concentration.

The CFD (Ansys-Fluent) calculations were compared with the results of the Gaussian model (Briggs model) on a flat surface, the calculations are carried out in two lines parallel to the buildings and taken at the center of the plume $y = 0$ m, with $x = 270$ m between the buildings and $x = 1200$ a little far from the buildings, and $z = 300$ for the two lines, they show good consistency (figure 8). In this case, spatial dependence of the parameters of Gaussian distribution was taken from the models of the first generation of Briggs. When simulating the flow through buildings exits, the effect of the significant increase in the concentration from leeward side of the building (figure 8) was found due to the turbulence created by the flow around the building.

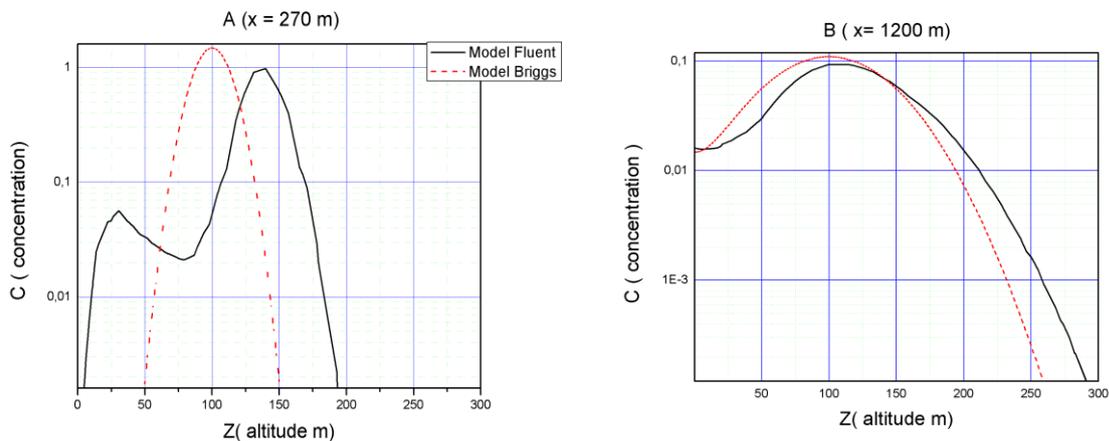


Figure 8. The vertical concentration profiles in lines A and B are indicated in Figure 7.

5. Conclusion

In this work a numerical study of the evolution of radioactive contaminant from an NPP ventilation pipe in the presence of obstacles is presented. This type of study allows the determination of areas where wind speed is accelerated by the presence of surrounding obstacles. The results essentially show that an obstacle in the path of a gas release can divert it either off course or/and from its direction; laterally or vertically. Near buildings, the concentration of radioactive emissions calculated by the Gauss model has a significant error.

References

- [1] Snyder W 1981 *Guidelines for Fluid Modelling of Atmospheric Diffusion*. EPA office of Air quality, planning and standards, Research triangle park, NC, EPA-600/8-81-009
- [2] Schulman L L and Scire J S 1993 Building downwash screening modeling for the downwind recirculation cavity. *J. Air. Waste. Manag. Assoc.* **43** 1122–27
- [3] Saathoff P, Gupta A, Stathopoulos T and Lazure L 2009 Contamination of fresh air intakes due to downwash from a rooftop structure *J. Air. Waste. Manag. Assoc.* **59** 343–53
- [4] Merone R N 2004 Wind tunnel and numerical simulation of pollution dispersion: a hybrid approach. Working paper, Croucher Advanced Study Institute on Wind Tunnel Modeling, (Hong Kong University of Science and Technology)
- [5] Blocken B, Stathopoulos T, Saathoff P and Wang X 2008 Numerical evaluation of pollutant dispersion in the built environment: comparisons between models and experiments *J. Wind. Eng. Ind. Aerod.* **96** 1817-31
- [6] Murakami S and Mochida A 1988 3-D numerical simulation of airflow around a cubic model by means of the k- ϵ model *J. Wind. Eng. Ind. Aerod.* **31** 283–303
- [7] Meroney R N, Leitl B M, Rafailidis S and Schatzmann M 1999. Wind-tunnel and numerical modeling of flow and dispersion about several building shapes *J. Wind. Eng. Ind. Aerod.* **81** 333-45
- [8] Hanna S R, Briggs GA and Hosker RP 1982 *Handbook on Atmospheric Diffusion*. DOE/TIC11223 (U. S. Department of Energy, Oak Ridge, TN)
- [9] Leroy C. 2010 A study of the atmospheric dispersion of a high release of krypton-85 above a complex coastal terrain, comparison with the predictions of Gaussian models (Briggs, Doury, ADMS4) *J. Environ. Radioact.* **101** 937–44
- [10] ANSYS FLUENT User's Guide.
- [11] Durbin PA and Petterson Reif BA 2001. *Statistical Theory and Modelling for Turbulent Flows* (John Wiley & Sons, Chichester, UK)