IOP Conference Series: Earth and Environmental Science

Flow and dispersion simulation using Computational Fluid Dynamics: A Case Study for EduCity in Iskandar Malaysia

To cite this article: Jo-Han Ng *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **268** 012129

Flow and dispersion simulation using Computational Fluid Dynamics: A Case Study for EduCity in Iskandar Malaysia

Jo-Han Ng[1,2,*], Shawn Navarednam[1], Kang Yao Wong[1,2] and Cheng Tung Chong[3]

[1] Faculty of Engineering and Physical Sciences, University of Southampton Malaysia (UoSM), 79200 Iskandar Puteri, Johor, Malaysia
[2] Energy Technology Research Group, University of Southampton, Southampton, SO17 1BJ, United Kingdom
[3] China-UK Low Carbon College, Shanghai Jiao Tong University, Lingang, Shanghai 201306, China
*Corresponding author email: J.Ng@soton.ac.uk

Abstract. Urbanisation has brought about various changes in the rural landscape, leading to a change in urban wind flow and pollutant dispersion. In this study, the flow and pollutant dispersion in a newly populated education hub, EduCity in Iskandar Malaysia are simulated using computational fluid dynamics (CFD). In order to ensure accuracy of the numerical simulation, Geographical Informational System (GIS) data was used to replicate the terrain of the region, while the main buildings of the education hub were measured to a high level of detail. The inlet boundary conditions utilised the power law to describe the reference velocity-height relationship, while sand grain roughness and density of vegetation were also taken into account. In order to validate the simulation, wind conditions at three different locations within EduCity were measured and the maximum deviation of 3.45% were observed across various location-time combinations. Upon validation of the integrated model, upstream wind velocity was varied, while pollutant profiles and wind directions were fixed for the urban flow and pollutant dispersion simulations. From the results, it is shown that upstream wind velocities greatly change the localised characteristics of wind and pollutant flows, although limited by the constraints of the physical configurations of the buildings. The results show that CFD simulations can be used to manage the air qualities in education hubs, and helps dictates the planning of urban development.

1. Introduction

Within an urban setting, the flow field around buildings can be difficult to predict due to the geometry of the structures, largely resulting in bluff-body flows. It is the presence of buildings and their associated dimensions that influence outdoor microclimates such as relative humidity, air pollution, and wind velocity [1]. This form of urbanisation will lead to the formation of the Urban Canopy Layer (UCL), which is a layer of air from the ground to the average height of buildings and trees [2]. It is predominantly affected by the terrain roughness and governed by the density of buildings. Within the UCL, the flow variables become increasingly inhomogeneous, pertaining to a large array of complications such as pedestrian wind speed, urban heat islands and pollutant dispersion. Thus, in recent years, computational fluid dynamics (CFD) has been used intensively to study the effects of urbanisation, especially in the lower part of the UCL. Many case studies of real locations have been carried out on both the urban macroscale and microscale (less than 2km). This would include a CFD study of the urban morphology in Beijing, coupled with the usage of geographic information system (GIS) where multiple building groups of similar scales were compared in efforts to estimate wind energy within each cluster [3]. Similarly, Chu *et al.* has carried out a pollution dispersion study in Hong Kong, modelling vehicular emissions as pollutant sources, showing the diffusion of pollutants for different wind speed and directions [4]. Recently, Kang *et al.* developed a CFD model to study the pedestrian wind comfort with consideration of tree drag parameterization in an urban setting, capable of reproducing the pollutant distribution pattern with respect to different tree leaf densities [5]. The building layout was plotted out using GIS data, ensuring that practical building heights and layouts were obtained. In spite of numerous CFD case studies in the past, there has been a lack in comprehensive research on the effects of urbanisation on educational hubs.

Here, a case study of EduCity, an education hub situated in the Iskandar Malaysia region, Johor, Malaysia, was done to investigate the effects of urbanisation by assimilating wind velocity and pollution dispersion studies, which could potentially lead to insights for the reduction of pollutant exposure during urban planning. Therefore, this numerical CFD simulation models stack pollutants across EduCity in Iskandar Malaysia using Gaussian dispersion model for the determination of gaseous emission flow.

2. Methodology

2.1. CFD modelling setup

Commercial CFD software, ANSYS Fluent 18.1 was used to perform a 3D transient state numerical simulation of air flow and pollutant dispersion. Table 1 summarises the submodels used in the flow and dispersion simulation. The two-equation turbulence model, renormalisation group *k*-epsilon (RNG k-ε) which solves for the turbulent kinetic energy (k) and its dissipation rate (ε), is coupled with species model using species transport. This combines both the 3D flow field and species transport equations to obtain the statistics for the pollutant dispersion, downstream of the light industrial zone.

Table 1. Submodels used in the integrated urban airflow and pollutant dispersion CFD simulation model.

Submodels	Type
Turbulence model	RNG k-ε
Species model	Species transport
Energy	Enabled

2.2. Computational domain and mesh generation

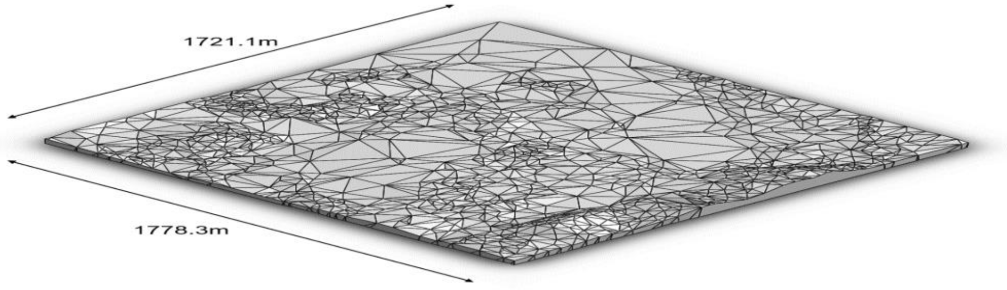

Figure 1. Dimension of meshed domain terrain without building models.

The urban model incorporates geographic informational system (GIS) to emulate actual conditions of the terrain. Elevation of terrain are accounted via a finite number of data points, containing packets of information to represent the irregular terrain features as shown in figure 1. Buildings within EduCity, were modelled to a high level of detail (LOD), through laser gauge measurements to reflect actual building dimensions, before they are simplified which provides the model with urban microscale (<2km) attribute. According to the Franke *et al.* [6], unnecessary wind acceleration can be avoided, as the simulation boundaries in the lateral and vertical directions conform to the minimum requirement of $8H_{max}$ and $5H_{max}$, respectively, with H_{max} equals to 50 m.

The full scale model was meshed with 1.85 million tetrahedral elements, prior to conversion to the face-based polyhedral mesh which reduces the number of elements to 0.45 million. The use of polyhedral meshes improves solver efficiency and solution accuracy, whilst also being apt at handling recirculating flows.

2.3. Boundary conditions

The cuboid-shaped simulated domain has a single area-based pollutants inlet, with the other boundary condition surfaces being pressure outlets and solid walls to represent open spaces and the ground, respectively. For the wind flow, the boundary inlet wind speed was assumed to obey the power law profile,

$$U(Z)=U_z\left(\frac{Z}{Z_z}\right)^{\alpha} \tag{1}$$

where U is the wind speed (m/s), Z is the height (m) and α is the power law exponent, which describe the terrain roughness effect on wind flow, in this case has been set at 0.198. The sand grain roughness and roughness constant value of the terrain to factor in the effects of surface roughness and foliage were obtained from Liu *et al.* [7].

3. Theory

3.1. Gas transport and diffusion modelling

In this study, a light industrial zone is modelled as an area source through summation of different point source emissions from their corresponding stacks. In order to determine the pollutant dispersion in urban area, five pollutants of interest are identified, namely, carbon monoxide (CO), nitrogen dioxide (NO_x), ground level ozone (O_3), sulfur dioxide (SO_2), and particulate matter of less than 10 microns in size (PM_{10}).

For the CFD simulation, all pollutants will be incorporated as a mixture with ambient air due to their gaseous nature, except for particulate-based PM. Therefore, a laser particle multi-functional detector with an accuracy of ±0.1 $\mu g/m^3$, is used to measure the ambient PM_{10} at the exact location from the wind velocity validation, allowing API to be calculated.

3.2. Holland's plume rise

The plume rise of a point source was predicted based on Holland's formula expressed as below [8],

$$\Delta h=\frac{V_s D}{u}\left[1.5+0.00268PD\left(\frac{T_s-T_a}{T_a}\right)\right] \tag{2}$$

where, Δh is the plume rise (m), V_s is the stack exit velocity (m/s), D is the diameter of the stack (m), u is the free stream wind velocity at stack height (m/s), P denotes pressure (millibars), T_s is the stack exit temperature (K), and T_a is the ambient temperature (K).

3.3. Gaussian area source plume model

The dispersion of an area source concentration field is based on the Gaussian distribution in both the horizontal and vertical directions, as plume travels away from the source, where concentration at any point (x, y, z) downstream of source is represented by [9],

$$c_{x,y,z} = \frac{Q}{2\pi\sigma_y\sigma_z u}\exp\left(-\frac{y^2}{2\sigma_y{}^2}\right)\left[\exp\left(-\frac{(z-h)^2}{2\sigma_z{}^2}\right)+\exp\left(-\frac{(z+h)^2}{2\sigma_z{}^2}\right)\right] \tag{3}$$

where Q denotes the source emission rate (g/s), h is the stack height (m), σ_y and σ_z are both the dispersion coefficients at cross-wind and vertical directions, respectively. The inlet profile of the area plume can then be described on the boundary layer using equations (2) and (3).

4. Model validation

The CFD model is validated against wind speed data across 3 distinct locations in EduCity, collected simultaneously and continuously using hand-held anemometers, over the period of 30 minute at 30 second interval. Longer periods of field measurement are not favourable due to variability of meteorological conditions [10]. Table 2 shows the modulus percentage error for each of the sample point against validation. The validation cases from simulation data points show good agreement with the field data set, with average error of 3.45%.

Table 2. Validation of field data set against simulated data points and their corresponding percentage error.

Modulus percentage error of sample points (%)			Average percentage error (%)
A	B	C	
1.71	8.14	0.5	3.45

5. Results and Discussion

The wind direction for the simulation is set to blow from North West towards EduCity, which corresponds to the downstream location of the light industrial zone. In this study, two different wind speed conditions of 0.9 ms^{-1} and 2.45 ms^{-1} were investigated, which is represented on the Beaufort wind force scale as light air and light breeze, respectively. The light air condition reflects the smoke drift threshold, while the light breeze condition allows wind to collide with obstruction, but not enough to raise dust from ground.

The pollutant concentration results and satellite view of the urban area studied were superimposed to show the interactions of urban structures and flow field.

5.1. *Low wind speed condition of 0.9 ms^{-1}.*

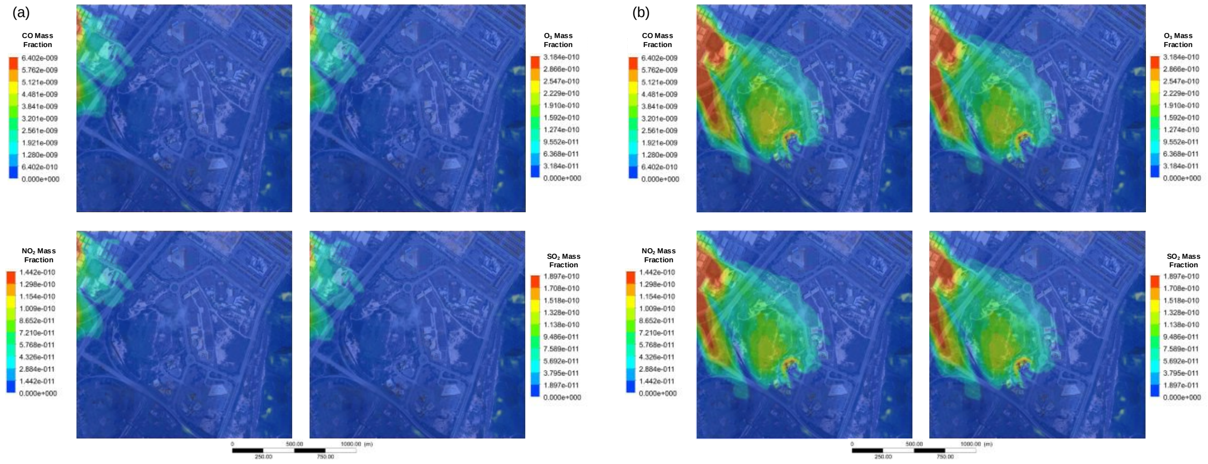

Figure 2. Pollutant concentrations of CO, O_3, NO_2, and SO_2 at 0.9 ms^{-1} wind speed condition at the real-time simulation of (a) 25 minutes (b) 50 minutes.

When wind speed is relatively low at 0.9 ms^{-1}, diffusion is a major mode of species transport for the gaseous pollutants, where flow is predominantly contributed by the concentration gradient between the source and the surrounding. Prior to the boundary of the simulation, a buffer zone was implemented to allow sufficient development of the main flow, which carries the mixture of air and pollutant.

Figure 2(a) shows the air dispersion of low wind speed condition, 25 minute into the real time from the boundary. The highest concentration of pollutant is CO, followed by NO_2, SO_2, and ground level O_3. Figure 2(b) describes the same wind condition at 50 minute into real time from boundary. The size of the pollutant distribution increased as the pollutant concentration field develops further downstream of the source, and grows from the initial diffusion pattern from Figure 2(a). Branching of the pollutants can be observed near the inlet of the airflow, due to the airflow being separated by larger structures. As the airflow travels further to the center of the simulated space, where the dominant high-rise structure is located, the concentration of pollutant increases significantly as accumulation occurred at the surface of these obstructions. On the contrary, the open field areas have relatively lower pollutant concentration as the pollutants are dispersed more evenly, reducing the individual pollutant mass fraction. Low wind speed condition indicates that higher pollutant concentration field is always closer to the source of pollution, where pollution eventually evolve downstream of the source. Higher concentration pollutant areas can also be found at high-rise surfaces perpendicular to the wind direction, which suggest that surface area that aligns to the dominant wind direction plays an important role in structural design, when wind comfort is part of the consideration.

5.2. High wind speed condition of 2.45 ms^{-1}.

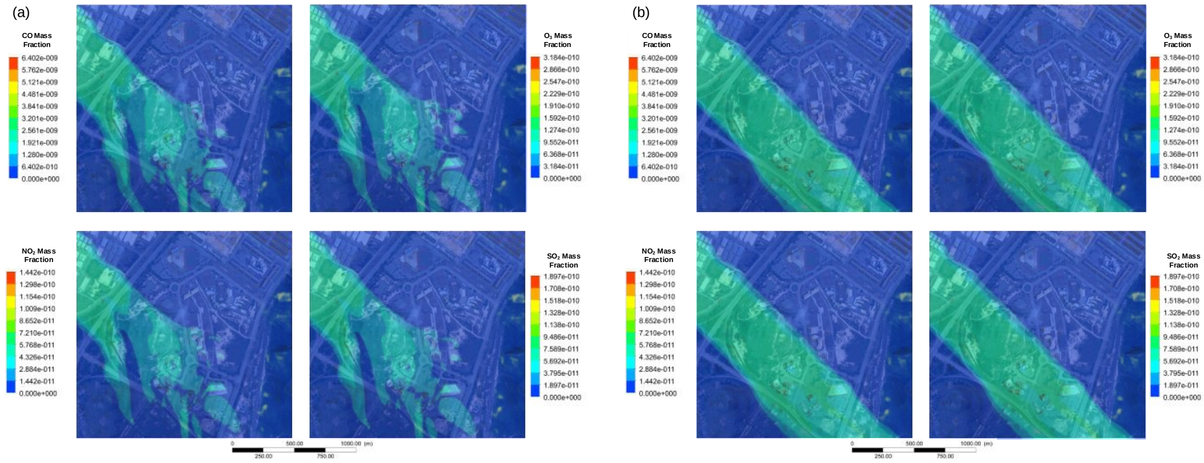

Figure 3. Pollutant concentrations of CO, O_3, NO_2, and SO_2 with 2.45 ms^{-1} wind speed condition at the real-time simulation of (a) 25 minutes (b) 50 minutes.

For the high wind speed condition of 2.45 ms^{-1}, the increase in the predisposition of turbulence can be observed from the airflow in Figure 3(a), which suggests that dilution of pollutant is greater. This is also reflected from the overall concentration level of pollutant, which accounts to about half to that of the low wind speed scenario. The high concentration level of pollutant accumulates at the high-rise structure, which also act as a bluff body in the airflow direction, while reduction in concentration is observable at the downstream locations for both 25 minute and 50 minutes into real-time.

The pollutant concentration converges into a single stream of airflow as time passes, which redistributes the pollutants rather evenly along the core pollutant stream. However, recirculation of airflow remained significant during the early development stages of the airflow, near the inlet boundary.

Conclusions

In this study, a 3D urban numerical simulation model of an education hub in Iskandar Malaysia was developed utilising a hybrid of geographical information system (GIS) data and actual measurements, to observe how urban airflow and gas dispersion affect the level of pollutant concentration. The simulation is conducted at two different windspeed condition, ranging from light air to light breeze conditions mimicking the prevailing wind conditions of the region. Through 3D CFD, the spatial-temporal air dispersion can be modelled to a high level of detail and accuracy, as fluid flow physics are taken into consideration. The results from the study can also used to identify the area with high accumulation of pollutants, where correlation with low air quality can be made. For both of the simulated wind speeds, pollutant with the highest peak concentration is CO, regardless of dispersion pattern, which suggest that flow physics is independent of pollutant dispersion for CO. However, SO_2 has a higher peak concentration in the low speed scenario, while ground-level O_3 is higher in mass fraction for high wind speed condition. Higher wind speed condition allows a greater dilution of air pollutant as compared to low wind movement, leading to less pollutant accumulation on the surface of structures. On the contrary, lower wind speed condition promotes spreading of pollutant with diffusion as a significant mode of transport, whereas pollutant of higher wind speed condition converges into a single stream. In all, the use of 3D CFD provides a tool for urban planners and policymakers to plan for the minimisation of pollutant exposures to inhabitants of newly populated regions.

Acknowledgement

The authors would like to acknowledge the support provided by the Malaysian Ministry of Education (MOE) under the Fundamental Research Grant Scheme (FRGS) through project FRGS/1/2016/TK07/USMC/02/2.

References

[1] Blocken B and Carmeliet J 2004 Pedestrian Wind Environment around Buildings: Literature Review and Practical Examples *Journal of Thermal Envelope and Building Science* **28** pp 107-159

[2] Rotach M W et al 2005 BUBBLE – an Urban Boundary Layer Meteorology Project *Theoretical and Applied Climatology* **81** pp 231–261

[3] Wang B, Sun S and Duan M 2018 Wind potential evaluation with urban morphology - A case study in Beijing *Energy Procedia* **153** pp 62–67

[4] Chu A K M , Kwok R C W and Yu K N 2005 Study of pollution dispersion in urban areas using Computational Fluid Dynamics (CFD) and Geographic Information System (GIS) *Environmental Modelling and Software* **20** pp 273–277

[5] Kang G, Kim J, Kim D, Choi W and Park S 2017 Development of a computational fluid dynamics model with tree drag parameterizations: Application to pedestrian wind comfort in an urban area *Building and Environment* **124** pp 209–218

[6] Franke J, Hellsten A, Schlunzen K and Carissimo B (2011). The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary. *International Journal of Environment and Pollution*, **44**(1/2/3/4), p 419

[7] Liu S, Pan W, Zhao X, Zhang H, Cheng X, Long Z. and Chen Q (2018). Influence of surrounding buildings on wind flow around a building predicted by CFD simulations. *Building and Environment*, 140, pp 1–10

[8] Brummage K (1968). The calculation of atmospheric dispersion from a stack. *Atmospheric Environment (1967)*, **2**(3), pp 197–224.

[9] Seinfeld J and Pandis S (1997). Atmospheric chemistry and physics: From air pollution to climate change. New York: Wiley-VCH

[10] Franke J, Hirsch C, Jensen A G, H, Krüs H W, Schatzmann M, Westbury P S, Miles S D, Wisse J A and Wright N G (2004). Recommendations on the use of CFD in wind engineering. J.P.A.J. van Beeck (Ed.), COST Action C14, Impact of Wind and Storm on City Life Built Environment. *Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics*, 5–7 May 2004, von Karman Institute, Sint-Genesius-Rode, Belgium.