Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 00 (2017) 000–000

Procedia Engineering

www.elsevier.com/locate/procedia

International High- Performance Built Environment Conference – A Sustainable Built Environment Conference 2016 Series (SBE16), iHBE 2016

CFD Methodology Development for Singapore Green Mark Building Application

Pao-Hsiung Chiu[a,*], Venugopalan S.G. Raghavan[a], Hee Joo Poh[a], Erna Tan[b], Osrithalita Gabriela[b], Nyuk-Hien Wong[b], T. van Hooff[c], B. Blocken[c], Ruixin Li[d], Su Ming Leong-Kok[d]

[a]*Institute of High Performance Computing, Agency for Science, Technology and Research (A*STAR), 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632, Singapore*

[b]*Department of Building, National University of Singapore, 4 Architecture Drive, Singapore 117566, Singapore*

[c]*Department of the Built Environment, Eindhoven University of Technology, P.O. box 513, 5600 MB Eindhoven, the Netherlands*

[d]*Green Mark Department, Building and Construction Authority of Singapore, No. 200 Braddell Road, Singapore 579700, Singapore*

Abstract

In the recent decade, investigation on the total building performance has become increasingly important for the environmental modelling community. With the advance of integrated design and modelling tool and Building Information Modelling (BIM) development, it is now possible to simulate and predict the building energy efficiency, air quality & health assessment, risk analysis & mitigation scenario for our urban planning analysis; all seamlessly in a single urban digital platform. In order to achieve the national goal of at least 80% of the buildings in Singapore to be green by 2030, Singapore Government has introduced the new BCA Green Mark 2015 scheme for accelerating the green building agenda. During the recent third Green Building Masterplan announced in 2015, it was decided to engage building tenants and occupants more actively to drive energy consumption behavioural change and to address the well-being of the people. Following up from this Masterplan, it is important for both the stakeholders and agency to jointly develop Performance Driven and Scientific Based Simulation Methodology and Evaluation Parameters as a frame work to evaluate the building design based on Singapore's hot and humid climate and densely-built-up urban areas for the Green Mark 2015 Scheme. In this paper, we will present the methodology and perform a baseline case study for the natural ventilation performance with the typical Non-Residential Building (NRB) industrial building. This can be resulted in the comprehensive CFD Quality Check List for the Environmental Sustainable Design (ESD) consultant in order to maintain modelling result accuracy. Demonstration on Indoor Air Quality (IAQ) using Air Exchange Effectiveness (AEE) as performance indicator will also be illustrated.

Keywords: CFD; Green Mark scheme; Environmental Sustainable Design; Singapore

* Corresponding author. Tel.: +65-6419-1356 ; fax: +65-6463-0200.
E-mail address: chiuph@ihpc.a-star.edu.sg

1. Introduction

It has become the definitive tendency of building an artistic type of high-rising buildings in almost all metropolitan cities due to commercial demands and rapid growth of population. These recently built modern artistic building shapes and ever-increasing heights pose their problems related to the wind effects on the building itself and also on its surroundings. Designing a high-rise green building in a densely built-up urban area in a tropical region like Singapore is a unique challenge. This motivated the Building and Construction Authority (BCA) of Singapore to launch the Green Mark (GM) scheme in 2005. This scheme provides an assessment methodology to drive the building designs to be more environment-friendly and sustainable. In order to more accurately assess the efficiency of natural ventilation, the Green Mark scheme began to include the Computational Fluid Dynamics (CFD) simulations in 2008. By incorporating CFD simulations, it is easier to investigate all the functional units of interest, and providing guidance for improving the efficiency of natural ventilation. However, airflow over high-rise buildings can be extremely difficult to predict. In an urban environment, wind is a phenomenon of great complexity due to its interaction with the building structures, vegetation, waterbody and highly heterogeneous mixed land usage. Multi-scaled and rotational eddies carried along the wind stream near ground surfaces and building structures can always result in gusts and/or have a highly turbulent structure. As a result, an accurate modelling of wind flow around buildings must take into account the turbulence modelling in order to have better prediction of the natural ventilation system. In recent years, the rapid development of computing power and ever-improving accuracy and efficiency of numerical methods have enabled large-scale simulations of the flowfield around a building with complex geometry and has been particularly useful in understanding the flowfield and its resulting dynamic loading on some common shapes of buildings. The emerging use of CFD tools, which are popular especially at the conceptual design stage for high-rising buildings, motivates the present study to improve the CFD methodology for Non-Residential Building (NRB) for the BCA Green Mark scheme 2015.

This paper is organized as follows. Firstly we have this introduction section, followed by section 2 to describe the governing equations and turbulence model. It is following by briefly describing the Numerical methods as well as boundary conditions for the commercially available finite-volume package ANSYS FLUENT V16.2. We also present the results in setction 3 for justifying the horizontal homogeneity of atmospheric boundary layer (ABL). This is followed by the discussion of results on natural ventilation design and the newly proposed Indoor Air Quality performance indicators for natural ventilation space, namely Air Exchange Effectiveness (AEE). Lastly, some concluding remarks will be made in section 4

2. Governing Equations and Numerical Model

To date, it is still too expensive to resolve all the turbulent details numerically for large-scale flow simulations. Therefore, the common practice is to solve the Reynolds-Averaged Navier-Stokes (RANS) equations together with the turbulence models to compute the averaged turbulent stresses. Within the incompressible flow context, appropriate conservation equations for modelling the viscous airflow around a group of buildings of complex shapes can be expressed as follows:

$$\nabla \cdot (\rho \vec{V}) = 0 \tag{1}$$

$$\nabla \cdot (\rho \vec{V} \times \vec{V}) = -\nabla p + \nabla \cdot (\mu_{eff} (\nabla \vec{V} + (\nabla \vec{V})^T)) \tag{2}$$

where ρ, t, $\vec{V}$, p and μ_{eff} represent the fluid density, time, fluid velocity vector, static pressure and fluid viscosity. Note that the above effective viscosity can be decomposed as the sum of the laminar viscosity μ_L and the turbulent viscosity μ_T, implying that $\mu_{eff} = \mu_L + \mu_T$.

Among the numerous models proposed in the past few decades for modelling the turbulent viscosity μ_T, both large eddy and two-equation models have been widely adopted to simulate the industrial and wind aerodynamics flows over bluff bodies. Since $k-\varepsilon$ two-equation model has been considered as the most reliable one with a reasonable accuracy and computational cost in predicting wind flows [1], we employ it in the present study to calculate μ_T by means of the Boussinesq eddy viscosity assumption given by $\mu_T = C_\mu \rho \frac{k^2}{\varepsilon}$, where k and ε represent the turbulent kinetic viscosity and the turbulent dissipation, respectively. The constant C_μ is set, as usual, to be 0.09. [2]. Within the context of $k-\varepsilon$

turbulence models, take standard $k-\varepsilon$ as an example, the transports of k and ε in the flowfield with a velocity vector $\vec{V}$ are governed by the following two respective differential equations, which are coupled to each other:

$$\nabla\cdot(\rho\vec{V}k)=\nabla((\mu_L+\frac{\mu_T}{\sigma_k})\nabla k)+G_k+G_b-\rho\varepsilon \tag{3}$$

$$\nabla\cdot(\rho\vec{V}\varepsilon)=\nabla((\mu_L+\frac{\mu_T}{\sigma_\varepsilon})\nabla\varepsilon)+\frac{\varepsilon}{k}(C_{\varepsilon_1}P_k+C_{\varepsilon_2}P_\varepsilon) \tag{4}$$

The constants in the above two coupled equations are specified with the values given by $C_{\varepsilon_1}=1.44$, $C_{\varepsilon_2}=1.44$, $\sigma_k=1$ and $\sigma_\varepsilon=1.3$ [2]. The turbulent production term shown in the source term of equation (3) is due to shear stress and is defined as $P_k=\mu_T\nabla\vec{V}(\nabla\vec{V}+(\nabla\vec{V})^T)-\frac{2}{3}\nabla\cdot\vec{V}(3\mu_T\nabla\cdot\vec{V}+\rho k)$.

While standard $k-\varepsilon$ turbulent model has the property of offering robustness, economy and reasonable accuracy, it doesn't perform well when applied to the problems involving non-equilibrium boundary layers. It tends to under-predict the onset of separation and influence the prediction performance, at least, for the flow over an obstacle of various shapes. To resolve this problem, the realizable $k-\varepsilon$ turbulent model [3], implemented in Ansys Fluen, is chosen in this study.

In this study, we aim to explore the flowfield inside the functional unit of buildings due to the approaching wind, to evaluate the performance of natural ventilations as well as AEE.

To predict the physically and geometrically complex flowfield, we employ the commercially available tools and link them to render a likely numerical simulation wind tunnel (NSWT). The pre-processor in the currently employed NSWT platform is Fluent Gambit. By virtue of the embedded transformation Gambit IGES file in Fluent, we can generate the required surface meshes, followed by the interior mesh. The mesh that we obtain is hexahedral, which aids in accurately resolving the boundary layer as well as in predicting flow separation and reattachment. The mesh is then exported to ANSYS Fluent V16.2 flow solver to solve for the primitive flow variables $\vec{V}$ and p, in addition to the two turbulent quantities k and ε.

Ansys Fluent solves within the control volumes for each working field variable in collocated grids distributed in the physical domain under investigation. The equations are discretized in a way to preserve the momentum and turbulent quantities k, ε in each of control volume. To fulfil the continuity of mass, the well-known segregated SIMPLEC (Semi Implicit Methodology for Pressure Linked Equations-Consistent) solution algorithm of Van Doormal and Raithby[4] is used for calculating the unknown pressure. With the updated pressure solution computed from the PDE equation, one can replace its value to improve the initially assumed or previously calculated pressure in the momentum equation to calculate the new velocity components. The iterative procedures described above will be terminated when the computed maximum difference for solutions $\vec{V}$, p, k and ε between the two sets of consecutive solutions, cast in $L_2-norms$, falls below the specified tolerance (10^{-4}). In Ansys Fluent, the input modulus enable us to preserve mesh arrangement for storing the field variables and convergence criteria. The boundary module is also provided for us to specify the boundary conditions for $\vec{V}$, k and ε. Nonlinear terms in the momentum equations will be linearized so as to render them into algebraic equations. To accelerate the solution processes, the algebraic multigrid scheme modulus will be chosen to solve the pressure from the Poisson equation. As for the solution solver for the derived algebraic equations for $\vec{V}$ from the momentum equation and the transport equations for k and ε, the AMG iterative solver is chosen to reduce the storage demand as well as to calculate the respective solutions $\vec{V}$ and (k,ε) efficiently.

The inlet boundary condition for the vertical wind profile is assumed to be given by the Logarithmic Law with reference height h to represent the atmospheric boundary layer (ABL), and determined by using the following equations:

$$U(z)=\frac{u^*_{ABL}}{\kappa}ln(\frac{z+z_0}{z_0}) \tag{5}$$

$$k(z)=\frac{u^{*2}_{ABL}}{\sqrt{C_\mu}} \tag{6}$$

$$\varepsilon(z)=\frac{u^{*3}_{ABL}}{\kappa(z+z_0)} \tag{7}$$

$$u^*_{ABL}=\frac{U_{ref}\kappa}{ln(\frac{h+z_0}{z_0})} \tag{8}$$

where u^*_{ABL} is the ABL friction velocity, $\kappa(= 0.42)$ is the von Karman constant, $C_\mu(= 0.09)$ is a constant, z_0 is aerodynamic roughness length, U_{ref} is the specified velocity at reference height h. In this study, the aerodynamic roughness length is set as 1m and reference height is chosen as 15.0m, together with table 1, to show the prevailing wind direction and speed. Note that velocity for the north wind case is used in this study.

Table 1. Tabulation of prevailing wind direction and speed obtained from National Eenvironment Agency of Singapore over a period of 18 years at reference height of 15m.

Wind direction	Mean speed (U_{ref}) (m/s)	Occurring period
North	2.0	December
North-East	2.9	March
South	2.8	June
South-East	3.2	September

In order to maintain the horizontal homogeneity of ABL, it is very important to define the ground boundary condition. For Ansys Fluent, The equation for the equivalent sand-grain roughness height k_s and corresponding roughness constant C_s can be expressed by [5]

$$k_s = \frac{9.793z_0}{C_s} \tag{9}$$

In this study, k_s is set as 0.24m, which implies C_s = 40.178.

3. Numercial results

In this study, Singapores typical Non-Residential Building (NRB) industrial building has been investigated with the newly proposed CFD Check list for the natural ventilation simulation. The objective of this check list is to:

1) To maintain CFD quality control for GM submission
2) To keep track of NRB Natural Ventilation Design Progress
3) To assist BCA GM administrative process (e.g. submission, revision, iterations)

The main items for the ventilation simulation check list including the following:

- Submission details: provide the project details for the submission.
- Building type: describe building functionality.
- Problem statement: objective and work scope.
- Site information: describe the site information and illustrate how the geometrical info is incorporated into the simulation model. The computational domain should be properly chosen so that the numerical error due to boundary effect can be minimized. In this study, we follow the guidelines as mentioned in [6], the directional blockage ratio for the target building and the computational domain have been set as 17%.
- CFD approach: what CFD solver is used. Describe the model assumption, limitations and geometrical simplification. As showed in section 2, steady three-dimensional incompressible Navier-Stokes fluid flow equations with realizable k-e turbulence model and standard wall function is used for this study.
- CFD meshing: describe what the mesh size is used for the CFD simulation. The mesh is set at 0.1 to 0.5m within the functional space of interest, 0.5 to 1.0m for building of interest and 1.0-5.0m for surrounding buildings. In the vertical direction, the mesh is set at 0.5-1.0m from ground surface to 10m height; followed by 1.0-5.0m to building height H_{max}. The total meshes are 7.79M.
- CFD boundary conditions: describe what the boundary condition is used and show the homogeneity of ABL. The inlet and ground boundary is set the same as described in section 2. For outlet boundary, pressure-outlet boundary condition is used (p = 0), while the side and top boundary is set as symmetric. All building walls are assumed as smooth walls.

- CFD numerical method: describe the numerical scheme and the convergence criteria that are used. Second order scheme is used for the momentum equations as well as the equations for turbulence kinetic energy and turbulence dissipation rate. For the details for the numerical settings, we summarize in table 2.

Table 2. numerical setting for the present study

variable	numerical scheme	relaxation factor	tolerance for AMG
pressure	standard	0.1	0.01
velocity	second order	0.5	0.1
turbulence kinetic energy	second order	0.6	0.1
turbulence dissipation rate	second order	0.6	0.01

- Documentation: full documentation of parameters.
- Design iteration: it should be have baseline case, followed by modified case and optimal case to show that CFD did indeed improve the design.

For the typical industrial buildings as the baseline case, shown in figure 1, we investigate the performance of natural ventilation on the functional spaces located at level 1 of the target buildings.

Before we analysis the simulated results, we investigate the horizontal homogeneity of ABL. As shown in figure 2, it indicate that the horizontal homogeneity of ABL is well established. Figure 3 shows the velocity contour for the building functional spaces. It can be seen that only the corridor has good natural ventilation; whereas in the other functional units, the wind speed and natural ventilation is rather poor. In this study, the Air Exchange Effectiveness (AEE) as performance indicator for Indoor Air Quality is also proposed and investigated for building natural ventilation system. AEE represents the ratio between shortest possible time needed for replacing the air in the room (τ_n) and the average of local values of age of air ($<\tau>$) as [7,8]

$$AEE = \frac{\tau_n}{<\tau>} = \frac{C_{out} Vol C_s}{C_{H=1.2} Vol C_s} = \frac{C_{out}}{C_{H=1.2}} \tag{10}$$

where C_{out} is area-averaged CO2 concentration at outlets, $C_{H=1.2}$ is area-averaged CO2 concentration at 1.2m height, Vol is volume of investigated unit and C_s is the pollution source. The value typically varies between 1 and 2; the higher value means the ability for air exchange is better in the functional unit of interest. In this analysis, we uniformly distributed the CO2 inside the functional unit of interest, which has been shown in figure 3, as volumetric source term and solve mass and momentum equations together with the species equation. After converged results for the species are obtained, we can estimated the AEE based on the simulated results. From the simulated results shown in figure 4, the present industrial building, AEE is only 1.1. It is not surprising due to the fact that the velocity is very low inside the functional unit. By the assessment of natural ventilation and AEE, it can be easily seen from present baseline case, the design for the functional units needs to be improved.

4. Conclusion

In this paper, we propose a new CFD Methodology for assessment of Green Mark ventilation simulation. By using the CFD checklist, all the important parameters like statements about the site information, the details about the mesh size as well as the numerical scheme and setting can be included, thus the quality and the accuracy for the CFD modelling can be achieved by following the present guideline. From investigating the practical baseline case of the typical Non-Residential Building (NRB) industrial building, we can analyse the simulated results and collect useful information such as wind velocity profile as well as turbulence intensity, to drive the natural ventilated building designs to be improved. The AEE is also been used as an indicator to know the effectiveness of air exchanging. Ultimately, it has been shown that through properly methodology development framework, CFD modelling can be used as scientific modelling tool to assess natural ventilation and IAQ performance for Green Building.

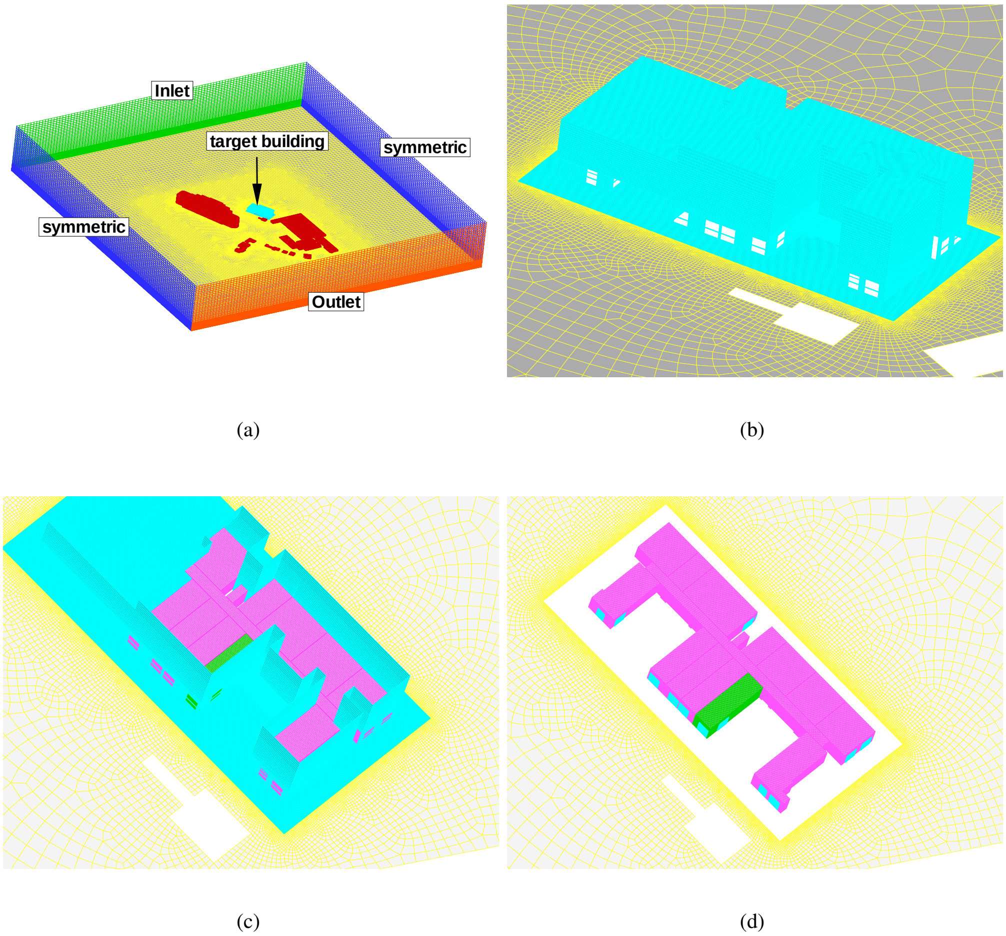


Fig. 1. Schematic for present investigating buildings. (a) investigated buildings; (b) the target building; (c) top view to show inside the functional units; (d) the functional units inside the target buildings.

Acknowledgements

This research is financially supported by BCA Research & Innovation Fund with vote number: 1.51.602.22153.00

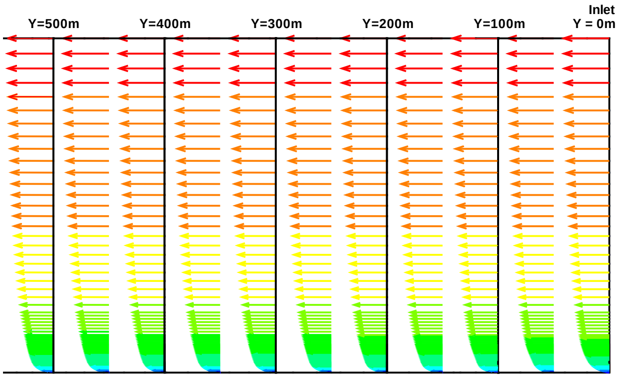
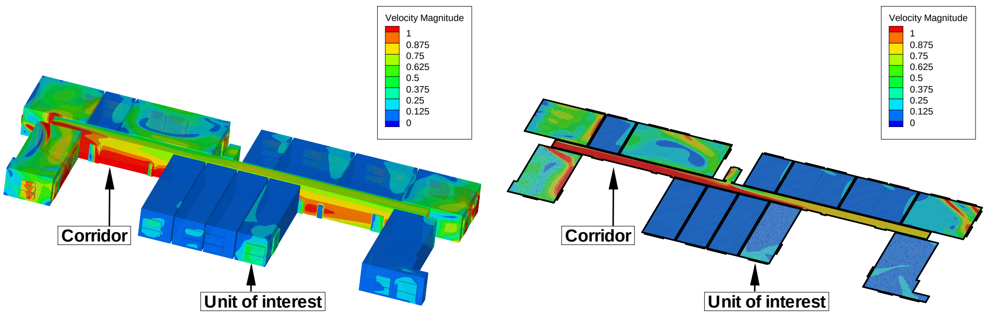
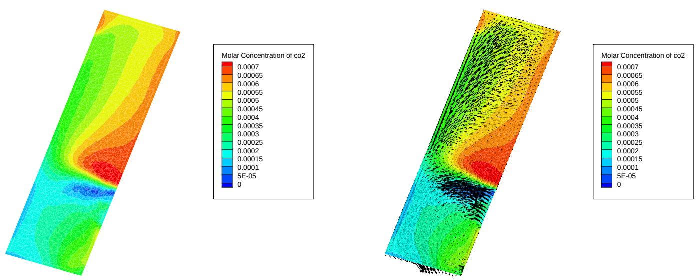


Fig. 2. Schematic to show the velocity vector at different cutting line.



(a) (b)

Fig. 3. Schematic to show the velocity contour for the functional spaces. (a) surface contour; (b)contour at 1.2m height.



(a) (b)

Fig. 4. Simulated results for the functional unit of interest. (a) contour for molar concentration of co2; (b) the same as (a) with velocity vector.

References

[1] D. A. Paterson, Computation of wind flows over three-dimensional buildings, Ph. D Thesis, The University of Queensland, 1986.

[2] B. E. Launder, D. B. Spalding, The numerical computation of turbulent flows, Computational Methods in Applied Mechanics and Engineering, 3 (1974) 269–289.

[3] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, A New k- ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids, 24 (1995) 227–238.

[4] J.P. Van Doormaal, G.D. Raithby , Enhancement of SIMPLE Method for Predicting Incompressible Fluid Flows, Numer. Heat Transfer, 7 (1984) 147–163.

[5] B. Blocken, T. Stathopoulos, J. Carmeliet, CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric Environment, 41 (2007) 238–252.

[6] B. Blocken, Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations, Building and Environment, 91 (2015) 219–245.

[7] A. Novoselac, J. Srebric, Comparison of Air Exchange Efficiency and Contaminant Removal Effectiveness as IAQ Indices, ASHRAE Transactions, 109 (2003) 339–349.

[8] D. Etheridge, M. Sandberg, Building Ventilation – Theory and Measurement. Wiley, 1996.