PAPER • OPEN ACCESS

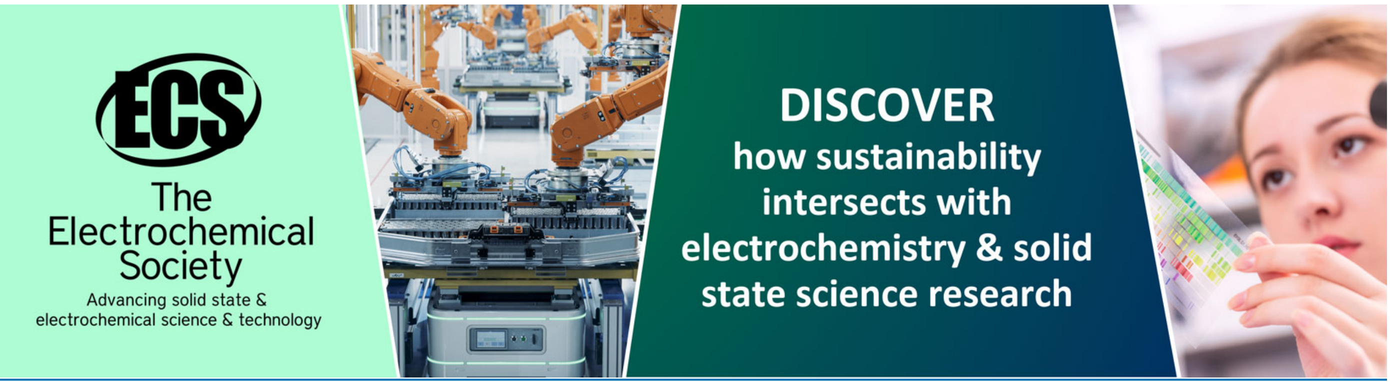
Development of GM2015 Computational Fluid Dynamics (CFD) Methodology for Naturally-ventilated Non-residential Buildings (NRB) in Singapore

To cite this article: Hee Joo Poh *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **238** 012079

View the article online for updates and enhancements.

You may also like

- Seismic vulnerability assessment of a steel-girder highway bridge equipped with different SMA wire-based smart elastomeric isolators
Farshad Hedayati Dezfuli and M Shahria Alam
- Contribution of vegetation and peat fires to particulate air pollution in Southeast Asia
C L Reddington, M Yoshioka, R Balasubramanian et al.
- Getting the numbers right: revisiting woodfuel sustainability in the developing world
Rob Bailis, Yiting Wang, Rudi Drigo et al.



This content was downloaded from IP address 18.224.44.100 on 14/05/2024 at 04:04

IOP Conf. Series: Earth and Environmental Science **238** (2019) 012079 doi:10.1088/1755-1315/238/1/012079

Development of GM2015 Computational Fluid Dynamics (CFD) Methodology for Naturally-ventilated Non-residential Buildings (NRB) in Singapore

Hee Joo POH[1,*], Pao-Hsiung CHIU[1], Chin Chun OOI[1], Venugopalan RAGHAVAN[1], Stephen WAN[1], George XU[1], Ruixin LI[2], Leong-Kok Su Ming[2]

[1] Institute of High Performance Computing, Agency for Science, Technology and Research (A*STAR), 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632
[2] Building and Construction Authority of Singapore, 52 Jurong Gateway Road, #12-01, Singapore 608550
[*] Corresponding author email: pohhj@ihpc.a-star.edu.sg

Abstract. This paper aims to share about the recent development on the performance-driven and scientific-based CFD quality control and framework in order to assess the naturally ventilated building designs in Singapore. The topic is an essential component for Green Mark 2015 Schemes, recently advocated by Building & Construction Authority (BCA), Singapore, for the sake of green and sustainable building designs. It covers six (6) different types of NRB buildings, namely with regard to industrial facilities, healthcare facilities, commercial atriums, hawker centres, sports facilities, and schools.. A comprehensive quality control checklist is also developed to ensure the quality and accuracy of the simulation results. In addition to the regulations from respective building stakeholders, the adopted parameters also take reference from internationally accepted standards and/or peer reviewed literatures. This CFD methodology development can help to supplement limited guideline on natural/passive ventilation for the building code of practice in Singapore. Finally, based on the efforts in this project, we will summarize the potential research topics that may require further collaboration with building practitioners, e.g. pilot scale study on certain building type and good modelling approach/tool to support both building designers and policy makers.

1. Introduction

Since CFD Ventilation Simulation Methodology and Requirement was included in the BCA Green Mark standard in 2005 (BCA, (2018)), there have been many projects undertaken for Green Mark assessment under this criterion. Throughout the years of the assessment evaluation process, it is found that the criteria used for Residential Buildings (RB) have been conveniently adopted for Non-Residential Buildings (NRB) with the passing criteria of area weighted wind velocity of 0.6 m/s.
Although air speed is an important factor that affects the occupants' comfort in the naturally ventilated buildings, thermal comfort of the occupants are usually determined by the combined effect of all six factors including (1) air temperature, (2) mean radiant temperature, (3) air speed, (4) humidity, (5) metabolic rate and (6) clothing insulation. The impact of each individual factor might be different for different building types. A set of dedicated Predicted Mean Vote (PMV)/Predicted Percentage of Dissatisfied (PPD) index for different types of buildings as the thermal comfort model would be

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

required to evaluate the natural ventilation design. In addition, for some naturally ventilated spaces such as industrial and sports facilities, the focus would turn to ensuring that the quality of the air is in good condition other than being thermally comfortable alone. Therefore, the development of assessment criteria for IAQ is required to assess the capability of the building design in delivering fresh air to the space of concern. Moreover, in order to cover the holistic range of building applications, specific CFD methodologies to cater for different types of building component/device are essential to ensure the accuracy of CFD simulation.

Other than that, Wind Driven Rain (WDR) penetration can be a problem for natural ventilated spaces, adversely affecting the building users and occupants and rendering these spaces "not useable" during times of heavy downpour. The lack of a WDR simulation methodology and evaluation criteria prevents building designers from conducting sufficient studies on their designs. Hence, a robust WDR simulation methodology and evaluation criteria will be developed through this study.

In view of the above, the development of Computational Fluid Dynamic (CFD) Simulation Methodology and Evaluation Parameters, Thermal Comfort Model and Air Quality Indices, as well as Simulation Methodology for WDR in Natural Ventilated Buildings of Non-Residential Buildings (NRB) for the BCA Green Mark 2015 Criteria is considered crucial and timely.

2. Literature Review

In the past, discussions with agencies and building stakeholders about CFD evaluation criteria and performance metrics on pilot projects in Singapore have been carried out (Su Ming, 2015). The parameters are listed as below:

1. Mesh independence studies
2. Atmospheric Boundary Layer simulation
3. Window modelling – any simplification?
4. Mechanical fan modelling – simplified model?
5. Heat source modeling (if any)
6. Louver modeling (simplified model)
7. Terrain modeling (resolution requirement)
8. Presence of greenery in urban landscape
9. Surface roughness effect on estate airflow
10. Baseline vs. Improved design comparison
11. Framework for architect & CFD modeler interaction
12. Surrounding buildings info to be obtained with reasonable accuracy
13. Performance indicator – ACH instead of average velocity

Globally, research community on urban ventilation has adopted internationally accepted standards and peer reviewed literature to ensure credible CFD results for building design and analysis. This include **domain size** that impose a maximum blockage ratio of 3% (Tominaga et al. (2008)), **grid mapping** to sufficient overall grid resolution and quality of the computational cells in terms of shape (including skewness), orientation and stretching ratio (Franke et al. (2007)), **roughness parameters** for accurate simulation of Atmospheric Boundary Layer (ABL) flow (Richards and Hoxey 1993, Blocken et al. 2007a, 2007b), **inflow boundary conditions** to express the vertical profiles for mean velocity and turbulence properties, **discretization schemes** with at least formally second-order accurate spatial discretization (Roache, (1997)), **iterative convergence criteria and oscillatory convergence** with a reduction of at least four orders of magnitude is recommended, Horizontal (in)homogeneity test, Grid-convergence analysis and lastly validation test cases. These are the important parameters for CFD natural ventilation study. Nonetheless, the application is still challenging to building designers without a proper and systematic checklist to guide the work flow and simulation results. Hence, it is important for Green Mark 2015 Criteria to develop checklist on CFD Ventilation Simulation.

IOP Conf. Series: Earth and Environmental Science **238** (2019) 012079 doi:10.1088/1755-1315/238/1/012079

3. Motivation and Objective

The objective of this project is to develop and implement GM2015 CFD simulation methodology and evaluation parameters to ensure good natural ventilation in six (6) non-residential building types. In addition, CFD working examples for modelling with Vegetation Canopy (for Aerodynamics Drag), Simplified Louver and High Volume Low Speed Fans in GM2015 methodology will also be developed and implemented.

Figure 1 below summarizes the six (6) NRB types, namely (a) industrial facilities, (b) healthcare facilities, (c) commercial atriums (d) hawker centres, (e) sports facilities, (f) schools which will be specified in new version of Green Mark 2015 criteria.

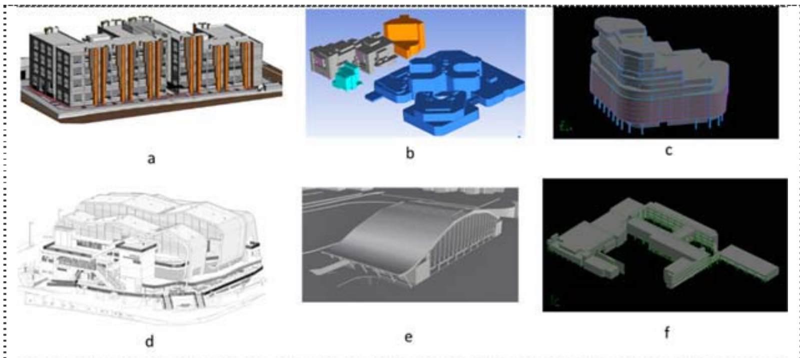


Figure 1. Development on GM2015 NRB CFD methodology for six (6) buildings types

4. Results & Discussion

Checklist on Ventilation Simulation

A comprehensive checklist for CFD simulation methodology and evaluation parameters to ensure good natural ventilation in six (6) non-residential building types has been developed and adopted for Green Mark 2015. This includes submission details, building types, problem statements, site information, CFD approach, domain, meshing, model, boundary conditions, numerical, user track record, documents and design iterations. More details can be obtained from BCA Non Residential Buildings NRB: 2015: Technical Guide and Requirements (BCA, 2015).

CFD Geometrical simplification

It is always necessary to simplify architectural model for CFD ventilation simulation. Indeed 70-80% of the effort is typically spent on this geometrical reconstruction and simplification. In our CFD simulation process, we have worked with building stakeholders to get the typical CAD files; working through the actual process and propose the geometrical simplification process. Figure 2 shows the typical files and processing tools used to produce the final CFD model, and we encourage building practitioner to follow this process closely for more efficient simulation work.

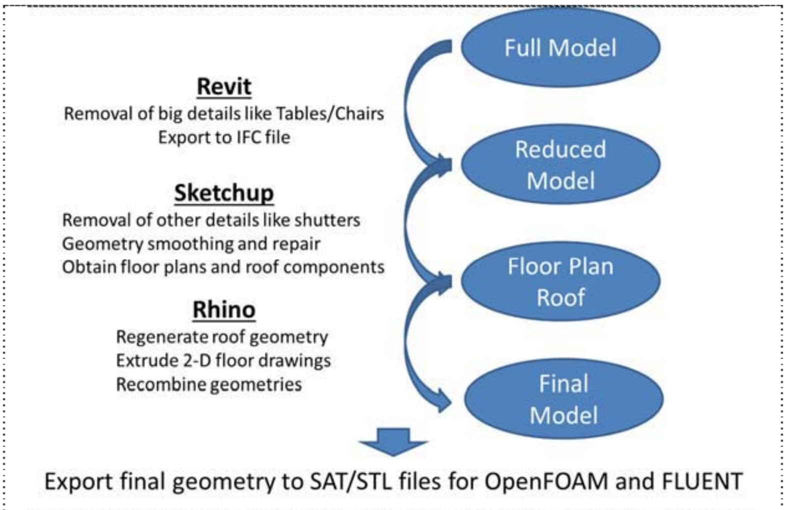


Figure 2. Geometrical Simplicaiton for CFD simulation process.

Computational Domain Decomposition and Meshing

It is important to ensure proper domain decomposition and high quality cells (e.g. hexahedral) being applied at the domain of interest, such as functional space and adjacent fluid domain. Figure 3 shows the commendable effort undertaken to generate hexahedral mesh within the functional unit and coupled with prismatic cells even in the presence of complex surrounding buildings.

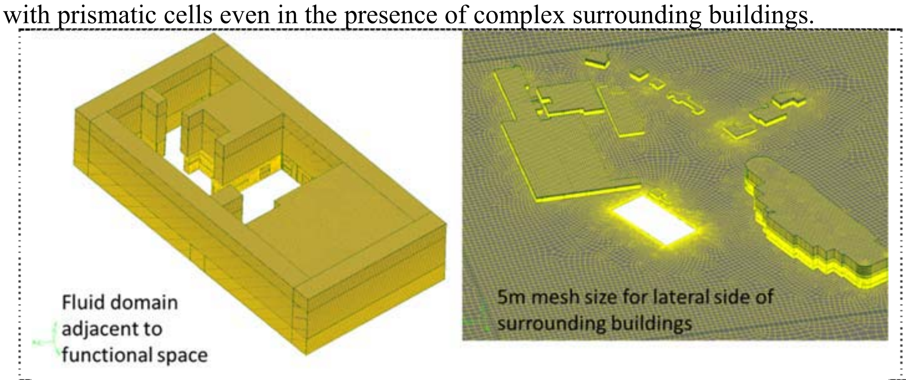


Figure 3. Proper domain decomposition to generate good quality mesh.

For the meshing typology and size distribution, we propose to use guideline as specified in Table 1 below:

Table 1: Guideline for mesh distribution and topology

Location	**Grid Size (m)**
Within the functional spaces of interest	0.1-0.5
Building of interest	0.5-1.0
Surrounding building	1.0-5.0
From ground surface to 10m height in vertical direction	0.5 – 1.0
Ten to H_{max} height in vertical direction, (H_{max} is the height of the tallest building among the group of buildings modeled explicitly)	1.0 – 5.0

Tree Canopy Modelling

In order to reproduce the tree aerodynamic effects on urban ventilation, we need to add extra terms to momentum and turbulence equations in order to mimic the decrease of wind velocity and increase of turbulence. This can be implemented with the appropriate Leaf Area Density (LAD) and leaf drag coefficient (C_D) values in the momentum sink term, as well as fraction of mean kinetic energy that is converted into wake turbulence kinetic energy (β_p) and short-circuiting of eddy cascade (β_d) in the turbulence source terms. The extra terms added to model equations are listed below. More details can be obtained from CFD working examples on tree canopy modelling prepared for BCA.

$$S_{u_i} = -\rho C_d (LAD) u_i U$$

$$S_k = \rho C_d (LAD)\left(\beta_p U^3 - \beta_d U k\right)$$

$$S_\varepsilon = \rho C_d (LAD) \frac{\varepsilon}{k}\left(C_{\varepsilon_4} \beta_p U^3 - C_{\varepsilon_5} \beta_d U k\right)$$

Figure 4 shows the typical velocity reduction obtained through tree canopy modelling implemented with the above formulation. It is indeed relatively easy to implement to different CFD solvers.

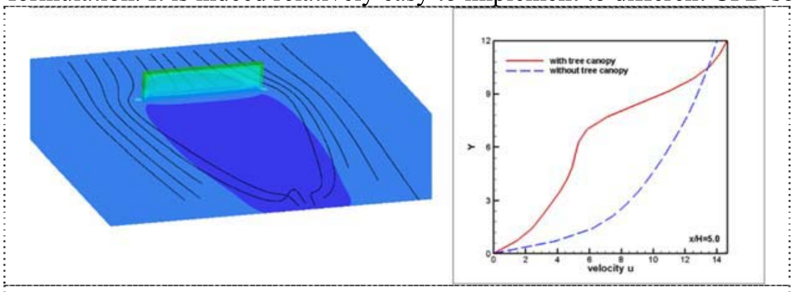


Figure 4. Velocity reduction with tree canopy modelling**.**

Louver Modelling

The porous media approach can provide a reasonable model for the louver at much reduced computational cost. We incorporate directionality by adding momentum source terms in the vertical direction but with loss of accuracy. The model is able to account for different louver lengths, louver angles, and air inflow velocity. The appropriate pressure drop coefficient and directional effect can be modelled through two stages. First stage involves modelling explicitly the louver profile with steady flow simulation in hypothetical room enclosure with a series of inflow velocities (1 – 10 m/s) in order to obtain the pressure drop coefficients across louver component. Second stage involves simplification by use of porous zone with the appropriate loss coefficients derived from the first stage to reproduce the directional effect; and subsequently perform the louver modelling implicitly on the actual functional unit. More details can be obtained from CFD working examples on louver modelling prepared for BCA. Figure 5 shows reasonable results is obtained for directionality and pressure drop when louvers are modelled with reduced model of momentum sink terms; while achieving high efficiency for simulation process with reduced computational cost.

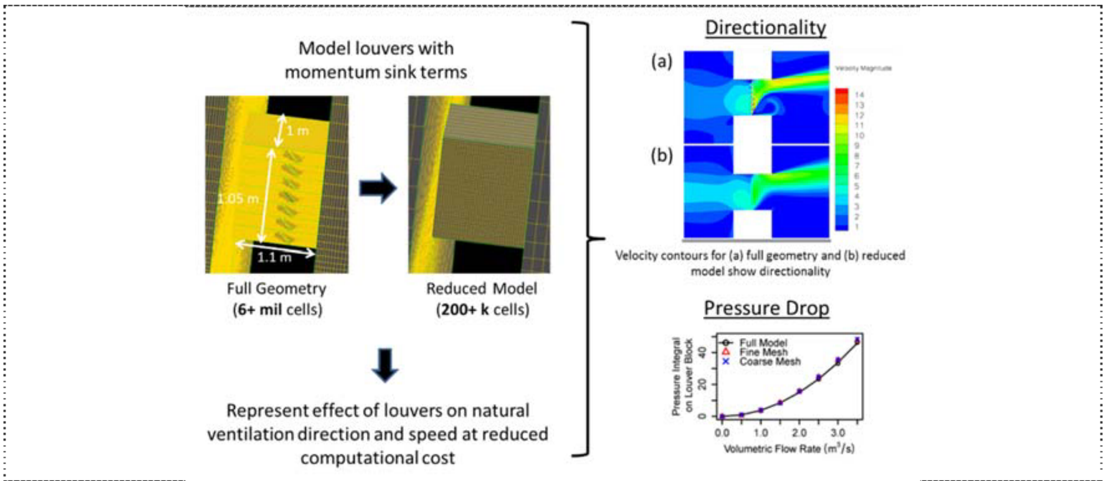


Figure 5. Louver Modeling with momentum sink terms.

<u>High Volume Low Speed (HVLS) Fans Modelling</u>
For HVLS fans modelling, the fan axial, swirl and radial flow field has to be captured correctly. It is proposed to employ multiple reference frame (MRF) method in transient mode to obtain a time-averaged flow-field (initial / starting conditions may be obtained from a steady state solution). Time step has to be at least 1/200 of the time taken for the fan to rotate one revolution (period of revolution). Convergence criterion: residuals have to be at least 0.001 or lower. For fan blades profile, at least five prismatic layers on each blade surface is recommended, to ensure adequate accuracy. Grid independent test is to be carried out. Higher order schemes for the convective terms are recommended (upwind scheme may be initially used to achieve stability). The fans are to be spaced such that recommended thermal comfort air velocities are achieved. Figure 6 shows the HVLS fans simulation results in which swirl flow components are captured in this MRF simulation; and this enables assessment of thermal comfort from air movement and extent of ventilation.

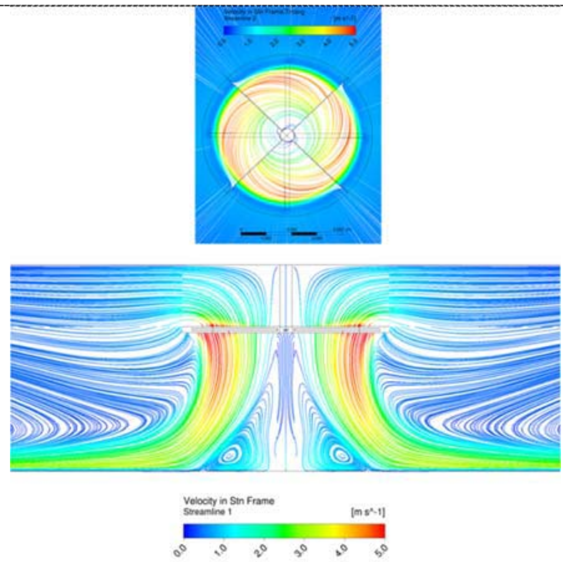


Figure 6. Example of a 20 ft HVLS fan simulation.

IOP Conf. Series: Earth and Environmental Science **238** (2019) 012079 doi:10.1088/1755-1315/238/1/012079

5. Conclusion

We have successfully developed Performance Driven and Scientific Based GM2015 CFD Simulation Methodology and Evaluation Parameters as a frame work to evaluate the Natural Ventilation building design based on Singapore's hot and humid climate and densely built-up urban areas for the NRB building type of healthcare facilities, industrial facilities, hawker centres, sports facilities, commercial atriums and schools. A comprehensive quality control checklist has also been developed and adopted to ensure the quality of the CFD simulation. In future, we hope to promote Simulation-based Decision-making in early building design stages so that biggest benefit on sustainable design and planning can be reaped at project upstream period and hence minimize the project cost.

References

[1] Building and Construction Authority Green Mark Scheme, https://www.bca.gov.sg/greenmark/green_mark_buildings.html (accessed in 22 Oct 2018).

[2] Personal communication with Leong-Kok Su Ming, BCA Greenmark Department Director, Apr 2015

[3] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T,. Yoshikawa, M., and Shirasawa, T., AIJ Guidelines for Practical Applications of CFD to Pedestrian Wind Environment Around Buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96, 1749 – 1761, 2008.

[4] Franke, J., Hellsten, A., Schlünzen, H., Carissimo, B. (Eds.). 2007. Best practice guideline for the CFD simulation of flows in the urban environment. COST Office Brussels, ISBN 3-00-018312-4.

[5] Richards, P.J., Hoxey, R.P. 1993. Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. *J Wind Eng Ind Aerodyn*, 46&47, 145-153.

[6] Roache, P.J. 1997. Quantification of uncertainty in computational fluid dynamics. *Annual Reviews in Fluid Mechanics*, 29, 123-160.

[7] Blocken B, Stathopoulos, T., Carmeliet, J, 2007a. CFD simulation of the atmospheric boundary layer: wall function problems. *Atmos Environ*, 41 (2), 238-252.

[8] BCA Non Residential Buildings NRB: 2015: Technical Guide and Requirements, https://www.bca.gov.sg/GreenMark/others/GM_NRB2015_Technical_Guide_Requirements_R3.pdf, (accessed in 22 Oct 2018)

Acknowledgments

The authors would like to acknowledge the financial support by Singapore Building and Construction Authority (BCA) Research and Innovation Fund with vote number: 1.51.602.22153.00 for this development work.