J. Basic. Appl. Sci. Res., 3(4)182-189, 2013

ISSN 2090-4304
Journal of Basic and Applied Scientific Research
www.textroad.com

DesignBuilder Verification and Validation for Indoor Natural Ventilation

Mohammad Baharvand[1], Mohd Hamdan Bin Ahmad[2], Tabassom Safikhani[3], Roshida Binti Abdul Majid[4]

[1,3,4] Department of Architecture, Faculty of Built Environment, Universiti Teknologi Malaysia, UTM Skudai, 81310 Johor, Malaysia.
[2] Institute Sultan Iskandar (ISI), University Teknologi Malaysia

ABSTRACT

Computer simulation has been used for simulating physical process and various researches have been conducted based on the computational fluid dynamics. Evaluating the accuracy of the simulation results is a significant point for users and developers. Verification and validation are the primary methods for responding to this issue. This paper examines the verification and validation of DesignBuilder as a simulation program for researchers and building designers. The focus of the study is on two main parameters of air velocity and temperature in natural cross-ventilation. Validation is examined comparing measurement data and computational fluid dynamics results. The comparison of the results indicates DesignBuilder can predict indoor temperature and air velocity with good accuracy and it can be used by researchers and designers to evaluate natural cross-ventilation.

KEYWORDS: DesignBuilder, CFD, Simulation, Verification, Validation, Natural cross-ventilation.

1. INTRODUCTION

More than 4 decades, engineers and researchers have been used computer to simulate physical process [23]. It provides a method for researchers to evaluate their ideas and designs in a fast and economical way. By this way, building designers can study and predict building performance in different aspects before construction. They can predict thermal comfort parameters of each space and optimize relative parameters to achieve better living conditions. Among these factors, ventilation and particularly natural ventilation is challenging due to various parameters which has an effect on airflow pattern.

Different methods have been proposed to study natural ventilation and air movement for indoor and outdoor of buildings. Chen [5] mentioned empirical models, small-scale experimental models, full-scale experimental models, multizone models, Zonal models and Computational Fluid Dynamics (CFD) models as a various approaches to examine natural ventilation. Among these methods, CFD is one of the most popular methods. Although CFD introduced for industrials purposes but now it has become a common method to evaluate ventilation and environment of buildings [1]. The applications of CFD models aren't limited to ventilation within buildings and it is known method for predicting different parameters of thermal comfort, indoor air quality, fire safety, HVAC system performance, etc. in different types of buildings [5]. CFD modelling is employed in the design process and it provides accurate and cost effective results [15; 17; 27;31]. It is a useful tool for engineers and designers to calculate inside and outside condition of buildings and acceptable results achieved in terms of energy usage and air flow based on CFD modelling [8].

Although CFD methods bring along many advantages but by developing the computer simulation and CFD usage, users and developers of the software have been faced new issue. They should know how they can rely on computer simulation and have a confidence in the simulation and CFD results. Verification and validation (V&V) are the primary methods for responding to the issue [23]. Oberkampf et.al [23] defined the verification and validation of computer simulation as below:

''Briefly, verification is the assessment of the accuracy of the solution to a computational model by comparison with known solutions. Validation is the assessment of the accuracy of a computational simulation by comparison with experimental data.'' Indeed verification deal with mathematical issue and validation is related to the real world and physic issue [23]. The American Society of Mechanical Engineers standard [2] defines verification and validation as below:

"Validation, which is defined as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model."and"Code verification establishes that the code accurately solves the mathematical model incorporated in the code."

The Computational Fluid Dynamics (CFD) numerically solves a set of partial differential equations for the conservation of mass, momentum (Navier–Stokes equations), energy, chemical-species concentrations, and turbulence quantities. The results of these calculations are various and it can indicate air pressure, air flow, air temperature distribution, relative humidity and etc. for inside and outside of the spaces [5]. These results lead to various application fields such as thermal comfort, fire safety, indoor air quality, HVAC system performance, etc. [5].

***Corresponding Author:** Mohammad Baharvand, Department of Architecture, Faculty of Built Environment, Universiti Teknologi Malaysia, UTM Skudai, 81310 Johor, Malaysia. E-mail: baharvand12@gmail.com Mobile: 0060-197113167

DesignBuilder is a CFD software which can calculate temperature, surface temperature, solar gain, radiant heating or cooling (low or high temperature), natural ventilation, mixed mode (natural and mechanical) ventilation, Displacement ventilation, thermal comfort level, energy and power costs with produce hourly reports of energy ,design load calculations to determine required HVAC equipment capacities, evaporative cooling, water use by occupants for cooking, cleaning or other uses, delighting (side lighting, skylights, or tubular daylight devices),Automatic interior or exterior lighting controls (such as occupancy, photocells, or time-clocks) and etc. In this paper verification and validation of DesignBuilder will be studied in terms of natural cross-ventilation prediction.

1 Turbulence Models Classification and Software Verification

Although the CFD method has been used more than 30 years to study natural ventilation, but engineers and researchers seeking for more accurate, reliable and faster CFD model [5]. Many parameters affect the accuracy of CFD results for natural ventilation study such as users' knowledge of fluid dynamics and experience and skill using numerical techniques. But the main parameter which has a critical effect on CFD results is appropriate selection of CFD approach and turbulence model [34].

Generally, turbulent flows are predicted in CFD by three methods: direct numerical simulation (DNS), large-eddy simulation (LES), and Reynolds-averaged Navier-Stokes (RANS) equation simulation with turbulence models[34].DNS features make it as an inappropriate method for indoor air distribution study at least for now due to it needs high grid resolution and very small time steps so powerful computer system should be provided to run CFD with DNS turbulence model[34].LES provides detailed information on instantaneous airflow and turbulence with the cost of still considerable computing time[34].To study air distribution within a space, the mean air parameters are more useful than instantaneous turbulent-flow parameters. Thus solving RANS equations is an interesting method which can predict air pattern fast [34]. RANS is a popular method for indoor air distribution due to it needs smaller requirements for computer resource and also user skills [34].

Zhai et.al [34] reviewed some of the most popular and useful turbulence models for indoor and also he looked at turbulence classification mentioned in his study (tab 1). As mentioned before selecting the correct turbulence model is important in the CFD process. Nielsen [21] studied about selecting a correct turbulence model. He founded that a simple correct turbulence model can be useful for provisional studies, a k-ε model for stratified flows, a low-Reynolds-number k-ε model for transport processes close to surfaces, and an LES model for providing the highest level of flow information [6].

Table 1. Popular and Prevalent Turbulence Models for Predicting Airflow in Enclosed Environment

Model Classification			Primary Turbulence Models Used in Indoor-Air Simulations	Prevalent Models Identified
RANS	EVM	Zero-equation	Zero-equation	
		Two-equation	Standard k-ε RNG k-ε Realizable k-ε	RNG k-ε
			LRN-LS LRN-JL LRN-LB	LRN-LS
			LRN k-ω SST k-ω	SST k-ω
		Multiple-equation	v2f-dav v2f-lau	v2f-dav
	RSM		RSM-IP RSM-EASM	RSM-IP
LES			LES-Sm LES-Dyn LES-Filter	LES-Dyn
DES			DES (S-A) DES (ASM)	DES-SA

1.1 Applications and Accuracy of Standard k-ε and EnergyPlus

The k-ε model family is the most popular turbulence model and it has the largest number of variants among mentioned turbulence models [34]. Standard k-ε, RNG k-ε and realizable k-ε model are some of these family members. Chen [4] compared five different models to predict indoor airflow of the simple room and he concluded standard k-ε and RNG k-ε can predict airflow pattern better. The selected software for this research is DesignBuilder, which uses EnergyPlus for simulation and standard k-ε turbulence model for CFD.

EnergyPlus is the official building simulation program of the United States Department of Energy, promoted through the Building and Technology Program of the Energy Efficiency and Renewable Energy Office [11]. This program is introduced by the Building Energy Software Tools Directory to predict energy simulation, load calculation, building performance, heat balance, and mass balance [11]. Crawley et al. [9] studied EnergyPlus, and compared it with previous programs in this field. He mentioned many advantages and benefits of EnergyPlus for building purposes. Also many studies have been used EnergyPlus in various fields [10; 13; 14; 20;25]. Zhai et al. [33] used EnergyPlus to calculate hybrid and natural ventilation in a building.

Additionally, many researches have been used standard *k*-ε for various purposes [7; 22; 28;30]. Some of these studies were especially considering on indoor air quality. They showed standard *k*-ε predicts indoor air distribution reasonably and well [12;19; 24; 29;32]. Cheung et al. [7] used standard *k*-ε for cross-ventilation validation study in a building and compare its results with a Jiang et al. [16] wind tunnel test. He found a well agreement of CFD and experimental results. Mistriotis et al. [18] checked the validation of the standard *k*-ε with experimental data for greenhouse in terms of natural ventilation and found good agreement of numerical method results and experimental test. Same results also gained by Bartzanas et al. [3]. He examined the validation of the standard *k*-ε in comparison with experimental results for indoor ventilation.

2 Software Validation

There are different methods to examine the validation of CFD code such as full scale experimental study, small scale experimental study, wind tunnel test, numerical methods and etc. Although full scale model and comparison of experimental measurements with CFD results is one the best way for CFD validation, but applying this approach for indoor study is not easy.

Graphical comparison between computational results and experimental data is the common validation method in CFD and “If the computational results ‘‘generally agree’’ with the experimental data, the computational results are declared ‘‘validated.’’’’[23]. There are some restrictions in experimental test which lead to some errors in the results. Expensive requirements and hardware, uncontrolled conditions, unmeasured environments and etc. are mentioned [23]. These factors make undesirable errors in experimental results.

The full-scale models in laboratory and in-situ measurements are very popular in ventilation performance studies. Most of the experimental measurements are done for generating data for CFD model validation, as they are very time-consuming and expensive [5]. Also the complexity of indoor airflow makes experimental investigation extremely difficult [5; 34]. To provide more accurate experimental data, researchers can use previous study and compare them with CFD results. Zhang et.al [35] examined different turbulent methods with some selected previous experimental data to compare the accuracy of turbulence models for internal airflow study.

2.1 Model Specifications

To evaluate DesignBuilder validation, experimental airflow and temperature study which has done by Stavrakakis et.al [26] in Greece was chosen to compare its measuring data with DesignBuilder simulation and CFD results. He examined indoor natural cross-ventilation experimentally and numerically, thus it can be an acceptable experimental test to evaluate software results in terms of natural cross-ventilation. The DesignBuilder version in this study is 3.1.0.080 Beta which uses EnergyPlus 7.2 for simulation.

The experimental chamber was located in the campus of the Technological Educational Institution of Chalkida in agricultural area of Psachna. It was in a flat rural environment surrounded by low-rise trees and low-rise residences at a sufficiently away distance. The test room dimensions were 6 m×4 m× 5.5 m (fig 1).

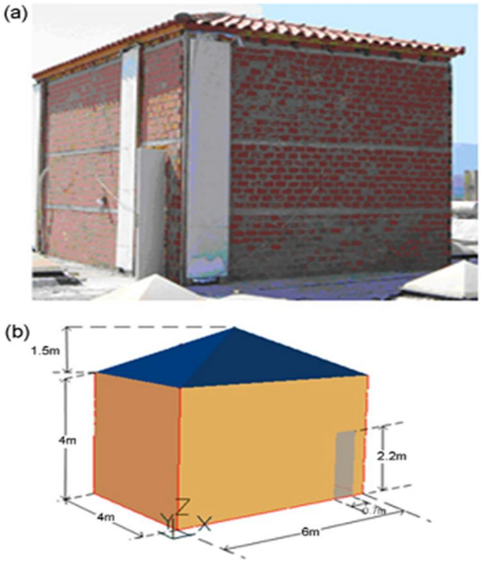


Figure 1. (a) Experimental test room and (b) Dimensions details

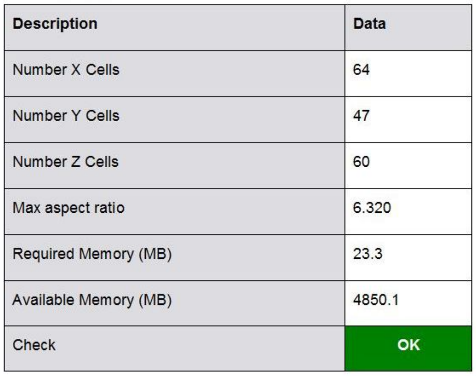
The surrounding walls are built from two series brick construction with a bubble material lamination among layers of aluminium foil placed in the 20 mm gap of the brick layers. The total wall thickness consists of 90 mm brick, 10 mm air gap, 1 or 2 mm reflective insulation, 10 mm air gap and 90 mm brick and roof covered with roman tiles and a radiant barrier reflective insulation system [26]. The experimental test was based on natural ventilation thus north wall and south wall had two doors which were completely open during the measuring and they were the only opening of the chamber. Small partition with 1m height was located attached to the north wall.

To provide accurate weather data during the study, the weather station was installed near the chamber to measure temperature, wind speed and wind direction. Stavrakakis et.al [26] main study was done for two typical summer days and temperature, air velocity, relative humidity and thermal comfort were examined during the test. Table 2 shows the selected measured weather data during the test. These data were measured at 19pm which is also selected for the CFD time in this study. Temperature and velocity inside the chamber was measured by KIMO thermo-anemometer multiprobes VT 200F with accuracy of ±3% for readings of 0–3 m/s and ±2% for reading of 0.1 8^{O}C.

Table 2. Weather Data Measured By Weather Station

Relative Humidity (%)	Outdoor Temperature (^{o}C)	Wind Speed at 7.5m (m/s)	Incidence Angle
24.8	32.35	2.85	90

The test room was modelled in DesignBuilder. The structure of CFD grid is uniform with 180480 cells include 64 cells on X axis, 47 cells on Y axis and 60 cells on Z axis with 6.32 as a maximum aspect ratio which is an acceptable ratio for modelling the test room (fig 2). The grid structure is optimized in monitoring points and near solid surfaces.



Description	**Data**
Number X Cells	64
Number Y Cells	47
Number Z Cells	60
Max aspect ratio	6.320
Required Memory (MB)	23.3
Available Memory (MB)	4850.1
Check	**OK**

Figure 2. Grid statistic

After providing grid structure, based on the experimental study, 6 monitoring points are defined within the room. Table 3shows the exact coordinate of these monitoring points.

Table 3. Monitoring Point Coordinate

Monitoring Points	B1	B2	B3	C1	C2
Coordinates (x, y, z)	3, 2, 0.2	3, 2, 2	3, 2, 3	0.7, 2, 0.5	0.7, 2, 1

2.2 Validation Results and Discussion

The simulation was run in the first step and in the second step simulation results were used as an input data for CFD. This ability is defined in DesignBuilder to achieve more accurate results in CFD. Figure 3 shows the measuring temperature results from experimental study and simulation results from DesignBuilder.

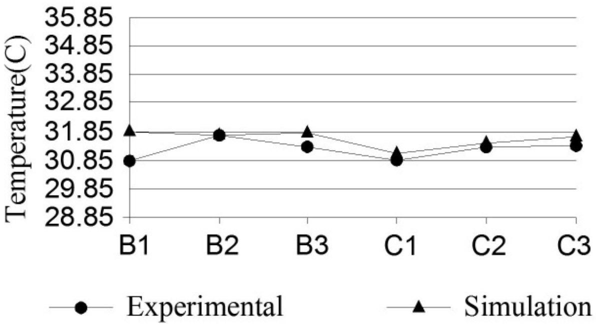


Figure 3. Experimental and simulation temperature results

Based on the figure 3 best agreements are achieved in B2 at the centre of the test room and the maximum difference observed in B1which is around 3%. Although there are some differences between results but all the CFD results are in a well agreement with experimental measurements. Figure 4 shows the temperature distribution in the middle of the test room.

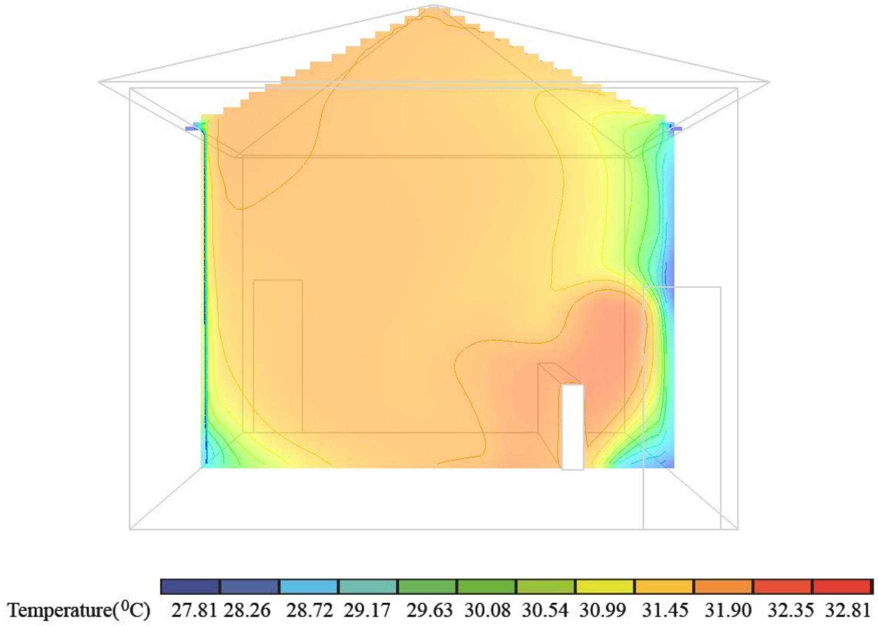


Figure 4. Temperature distribution in the middle of the test room

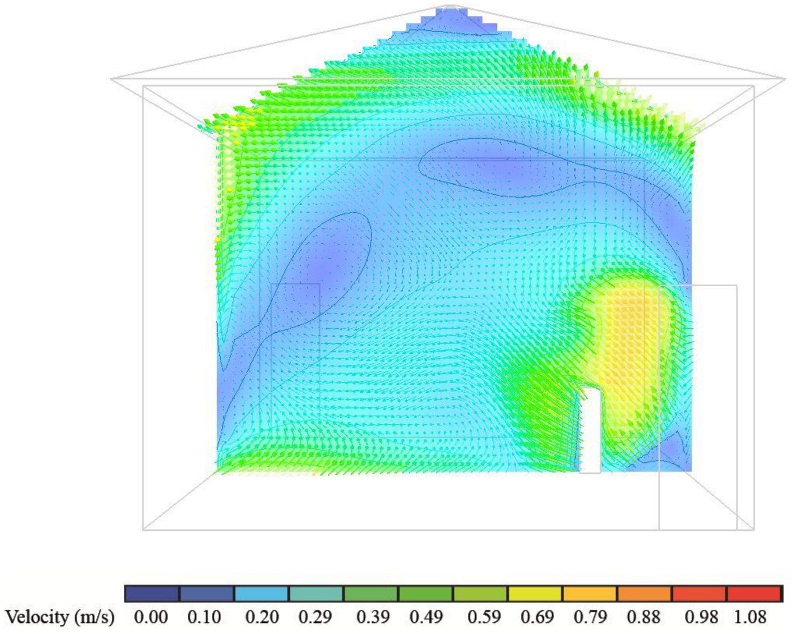


Figure 5. Air flow pattern in the middle of the test room

In the next step wind pattern was examined and experimental data compared with CFD air velocity results. Figure 5 shows the air distribution pattern within the test room. It shows the better airflow at the top and near the internal partition which can be due to the airflow angle from the south door. Figure 6 indicates B1, C1, C2 and C3 air velocity CFD results are in a well agreement with measuring results of experimental study.

As mentioned in the previous section, although experimental measuring is the best way for validation study, but it is a difficult approach in comparison with the other methods because of uncontrollable conditions, measuring tools errors, unmeasured parameters and etc. Change of wind speed and direction over time as well as significant surrounding buildings effects on the ventilation rates through building openings, form a very challenging for natural ventilation design [5]. These are can be the main reason of some differences in B2 and B3. Additionally Stavrakakis et.al [26] mentioned the infiltration parameter through small cracks and the existence of small obstacles such as packets used for the equipment storage which can affect the experimental results and cause some errors. Thus although some differences are observed in B2 and B3 but they can be in an acceptable range.

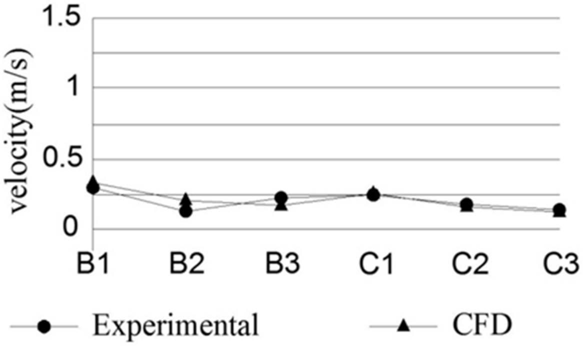


Figure 6. Experimental and simulation air velocity results

1 Conclusion

Indoor natural ventilation prediction is one the difficult study in buildings due to the number of parameters which has an effect on its pattern. Although simulation and CFD are the popular method to predict building performance in various aspects but the accuracy of the results should be checked. DesignBuilder is one of these software which can calculate different factors inside and outside of the buildings. Validation and verification of DesignBuilder were studied in this paper. The main engine of simulation in DesignBuilder is EnergyPlus and results accuracy of EnergyPlus was studied. In CFD section, DesignBuilder uses standard *k*-ε turbulence model which can predict airflow pattern and temperature distribution with an acceptable and reliable results. To examine the validation of the software, experimental test which was done by Stavrakakis et.al [26] was selected. Two main parameters of air velocity and temperature were examined in 6 monitoring points. Although there are some errors in the results but the results of software are in a well agreement with measuring data in experimental test. Finally it is concluded that DesignBuilder is a suitable simulation and CFD software to predict natural cross-ventilation.

Acknowledgments

The authors would like to acknowledge the research funding by Universiti Teknologi Malaysia (UTM), Ministry of Higher Education (MOHE) through Research University Grant (GUP). Also the authors would like to thank Dr. George Michael Stavrakakis from National Technical University of Athens for cooperation in experimental data.

REFERENCES

1. Asfour, O. S. and Gadi, M. B., 2007. A comparison between CFD and Network models for predicting wind-driven ventilation in buildings. Building and Environment, 42, 4079-4085.
2. ASME, 2009, USA: The American Society of Mechanical Engineers.
3. Bartzanas, T. et al., 2004. Effect of Vent Arrangement on Windward Ventilation of a Tunnel Greenhouse. Biosystems Engineering, 88, 479-490.
4. Chen, Q., 1995. COMPARISON OF DIFFERENT k-ε MODELS FOR INDOOR AIR FLOW COMPUTATIONS. Numerical Heat Transfer, Part B: Fundamentals, 28, 353-369.
5. Chen, Q., 2009. Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment, 44, 848-858.
6. Chen, Q. and Srebric, J., 2002. A procedure for verification, validation, and reporting of indoor environment CFD analyses. HVAC and R Research, 8, 201-216.

7. Cheung, J. O. P. and Liu, C.-H., 2011. CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings. Energy and Buildings, 43, 1149-1158.
8. Chien, C. H. et al., 2011. Buoyancy Ventilation Efficiency Analysis of a Conference Hall. Applied Mechanics and Materials, 71, 2442-2446.
9. Crawley, D. B. et al., 2001. EnergyPlus: creating a new-generation building energy simulation program. Energy and Buildings, 33, 319-331.
10. Eskin, N. and Türkmen, H., 2008. Analysis of annual heating and cooling energy requirements for office buildings in different climates in Turkey. Energy and Buildings, 40, 763-773.
11. Fumo, N. et al., 2010. Methodology to estimate building energy consumption using EnergyPlus Benchmark Models. Energy and Buildings, 42, 2331-2337.
12. Gadgil, A. J. et al., 2003. Indoor pollutant mixing time in an isothermal closed room: An investigation using CFD. Atmospheric Environment, 37, 5577-5586.
13. Guo, W. and Nutter, D. W., 2010. Setback and setup temperature analysis for a classic double-corridor classroom building. Energy and Buildings, 42, 189-197.
14. Henninger, R. H. et al., 2004. Analytical and comparative testing of EnergyPlus using IEA HVAC BESTEST E100–E200 test suite. Energy and Buildings, 36, 855-863.
15. Ji, Y. and Cook, M. J., 2007. Numerical studies of displacement natural ventilation in multi-storey buildings connected an atrium. Building Services Engineering Research and Technology, 28, 207-222.
16. Jiang, Y. et al., 2004. Validation of CFD simulations for natural ventilation. International Journal of Ventilation, 2, 359-369.
17. Laouadi, A. and Atif, M. R., 1998. Comparison between computed and field measured thermal parameters in an atrium building. Building and Environment, 34, 129-138.
18. Mistriotis, A. et al., 1997. Computational analysis of ventilation in greenhouses at zero- and low-wind-speeds. Agricultural and Forest Meteorology, 88, 121-135.
19. Nahor, H. B. et al., 2005. CFD model of the airflow, heat and mass transfer in cool stores. International Journal of Refrigeration, 28, 368-380.
20. Ng, L. C. et al., 2012. Multizone Airflow Models for Calculating Infiltration Rates in Commercial Reference Buildings. Energy and Buildings,
21. Nielsen, P. V., 1998. The selection of turbulence models for prediction of room airflow. TRANSACTIONS-AMERICAN SOCIETY OF HEATING REFRIGERATING AND AIR CONDITIONING ENGINEERS, 104, 1119-1127.
22. Norton, T. et al., 2010. Optimising the ventilation configuration of naturally ventilated livestock buildings for improved indoor environmental homogeneity. Building and Environment, 45, 983-995.
23. Oberkampf, W. L. and Trucano, T. G., 2002. Verification and validation in computational fluid dynamics. Progress in Aerospace Sciences, 38, 209-272.
24. Rouaud, O. and Havet, M., 2002. Computation of the airflow in a pilot scale clean room using K- ε turbulence models. International Journal of Refrigeration, 25, 351-361.
25. Sailor, D. J., 2008. A green roof model for building energy simulation programs. Energy and Buildings, 40, 1466-1478.
26. Stavrakakis, G. M. et al., 2008. Natural cross-ventilation in buildings: Building-scale experiments, numerical simulation and thermal comfort evaluation. Energy and Buildings, 40, 1666-1681.
27. Tan, G. and Glicksman, L. R., 2005. Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. Energy and Buildings, 37, 1049-1057.
28. Teitel, M. et al., 2008. Effect of wind direction on greenhouse ventilation rate, airflow patterns and temperature distributions. Biosystems Engineering, 101, 351-369.
29. Walsh, P. and Leong, W., 2004. Effectiveness of several turbulence models in natural convection. International Journal of Numerical Methods for Heat & Fluid Flow, 14, 633-648.
30. Wang, L. and Wong, N. H., 2008. Coupled simulations for naturally ventilated residential buildings. Automation in Construction, 17, 386-398.

31. Wang, X. et al., 2009. Mathematical modeling and experimental study on vertical temperature distribution of hybrid ventilation in an atrium building. Energy and Buildings, 41, 907-914.
32. Yang, T., 2004. CFD and field testing of a naturally ventilated full-scale building. University of Nottingham.
33. Zhai, Z. et al., 2011. Assessment of natural and hybrid ventilation models in whole-building energy simulations. Energy and Buildings, 43, 2251-2261.
34. Zhai, Z. et al., 2007. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1 - Summary of prevalent turbulence models. HVAC and R Research, 13, 853-870.
35. Zhang, Z. et al., 2007. Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 2—Comparison with Experimental Data from Literature. HVAC&R Research, 13, 871-886.