VOL. 12, NO. 10, MAY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

CFD INVESTIGATION OF INDOOR HYGROTHERMAL PERFORMANCE IN ACADEMIC RESEARCH STORAGE ROOM: MEASUREMENT AND VALIDATION

Majeed Olaide Oladokun[1], Maisarah Ali[2], Samsul Bahrin Osman[2], Niza Samsuddin[3], Hairul Aini Hamzah[4] and Md Noor Salleh[5]
[1]Department of Architecture, Building Services Engineering, International Islamic University, Malaysia, Kuala Lumpur, Malaysia
[2]Department of Manufacturing and Material Engineering, IIUM, Kuala Lumpur, Malaysia
[3]Department of Biomedical Science, IIUM, Kuantan, Malaysia
[4]Department of Basic Medical Sciences, IIUM, Kuantan, Malaysia
[5]Department of Biotechnology Engineering, IIUM, Kuala Lumpur, Malaysia
E-Mail: lyday011@yahoo.com

ABSTRACT

Poor hygrothermal performance exacerbates deterioration risk from mould growth, corrosion and damage to archival materials. Improved microcomputers' computational power has significantly advanced computational fluid dynamics (CFD) models and research developments in indoor airflow, heat transfer and contaminant transport. Nevertheless, numerous uncertainties exist in the CFD experiments which require adequate clarifications for improved results' reliability. This paper presents the measurement and validation of a CFD model for the investigation of the hygrothermal performance in an indoor environment with known cases of microbial proliferations. The room, 5.2 m × 4.8 m × 3.0 m high, is air-conditioned and ventilated by constant air volume (CAV) system controlling the indoor airflow and hygrothermal profiles with ceiling mounted four-way supply diffuser and extract grille for indoor air distribution. The methodology combines in-situ experiment and numerical simulation with a commercial CFD tool using the standard k–ε model. Microclimate and airflow parameters obtained from in-situ experiments were used as boundary conditions in the CFD. The study shows a good agreement between the predicted and measured indoor hygrothermal profile with less than 10% deviation. The results indicate that the model can be employed for further investigation with high confidence.

Keywords: CFD simulation, in-situ experiments, indoor climate, hygrothermal performance, uncertainty assessment.

INTRODUCTION

The need for energy efficiency in building design, construction and operation has continuously received considerable attention within the past few decades. This came as a result of large energy consumption evidence by the building sector which stands at about 40% of the global energy utilisation and contributing up to nearly 40-50% of the world carbon emissions [1, 2]. It had been previously documented [1, 3-5] that the Heating, Ventilating and Air Conditioning (HVAC) systems practically dominate the building sector energy consumption with over 60% of the total utilisation. With the energy efficiency needs on one hand comes the need to provide a healthy and comfortable indoor environment on the other. Despite the high energy consumption, the HVAC systems often found to result in sick building syndrome (SBS), building related illness (BRI) poor hygrothermal performance, and other indoor air quality (IAQ) related issues to the building occupants[6].Human being spend large part of their time in indoor environment (academic, health, recreational, commercial, etc.). The energy efficiency need, the comfort requirements and occupant longer stay indoor lead to a conflict between energy efficiency improvement and creation of healthy and comfortable indoor environment [7].

The built environments, over the past few decades, have therefore witnessed the emergence of building performance diagnostics. Building performance assessments are executed on the existing buildings in retrofit upgrades as well as new ones even before they are built. Various approaches exist for building performance appraisal [8, 9]. The authors classified such assessment approaches into analytical and empirical, numerical simulation (zonal models, multi zone models, computational fluid dynamics models, etc.) as well as experimental measurements (small-and-full-scale).While experimental approach is found most significant as it generates validation data for analytical and numerical simulation models, field experiments are expensive in terms of cost and access to free houses for in-situ experiments [10].

The rapid improvement in computer power in the past two to three decades has significantly influenced the development in computational models and progress in fluid dynamics research [11]. According to the authors, there is a change in speed from about 10^9 Flops in 1984 to 10^{13} Flops in 2002, an evidence that is corroborated by the observation of Li and Nielsen [12]. Despite these, numerous uncertainties exist in the in-situ and numerical experiments. It is therefore, pertinent to adequately clarify these uncertainties for improved quality of the obtained results. With good uncertainty clarifications, interpretation of results can be correctly done thereby increase the reliability of obtained results. Experimental errors or uncertainties can arise from the experimental set-up, data collection process and measuring equipment [13]. Detailed of a statistical approach of assessing instrumentation

ARPN Journal of Engineering and Applied Sciences

uncertainty for field experiment is presented in a recent work by the author [14].

Similarly, numerical experiments include error sources as modelling simplifications, mathematical and numerical models, and boundary conditions [11, 15]. Standard procedures on the execution and reporting of numerical experiments have evolved to reduce such uncertainties [16-18]. Benchmarking selected code with a previously documented numerical and experimental solution (verification) is recommended for ascertaining uncertainty from the simulation code while correlating simulated results with the measured values (validation) to ensuring that the results obtained from simulation are less prone to errors. For errors and uncertainties due to computational grid, grid dependency analyses remain the industry standards for improving discretisation accuracy.

Aims and objectives

This study presents the measurement and validation of a CFD model for the investigation of airflow and hygrothermal distribution in an indoor environment with known cases of microbial proliferations. The aim is to develop an error free model for the investigation of the indoor hygrothermal profile leading to indoor mould proliferation. The study objectives include: (1) to provide information on measurement procedures for indoor hygrothermal performance assessment, (2) to provide boundary conditions for the computational simulation of the facility, (3) to validate the modelling and simulation in accordance with best practice guidelines.

Methodology

Experimental set-up: The storage facility and ventilation system

The storage facility (Figure-1), 5.2 m × 4.8 m × 3.0 m high, is air-conditioned and ventilated by a constant air volume (CAV) air handling unit (AHU) that controls the airflow and hygrothermal distribution within the storage facility. The air distribution is a mixing ventilation type that consists of a ceiling mounted four-way square supply diffuser (600 mm x 600 mm) and a rectangular extract grille (600 mm x 300 mm). The lighting system consists of six numbers ceiling mounted fluorescent fittings (600 mm x 1200 mm) with two numbers 36W lamps. The air outlets as well as light fittings were installed to flush with the ceiling surface.

Measurement procedures

The measurement protocol was in two manifolds: steady state and time-series. Temperature, humidity and airflow data from the steady state measurements were use as input boundary conditions in the CFD simulations while time-series data for model validation.

Table-1. Specifications of measurement equipment.

Equipment	Variable	Accuracy	Resolution	Range
ALNOR AVM440 Airflow Instrument	RH	± 3%	0.1%	5 to 95%
	T	± 0.3°C	0.1°C	-10 to 60°C
	V	± 3% or ± 0.015m/s	0.01m/s	0 to 30m/s
Testo Quicktemp 860-T1	T	± 0.75% or ±0.75°C	0.1°C	-30 to 900°C

VOL. 12, NO. 10, MAY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

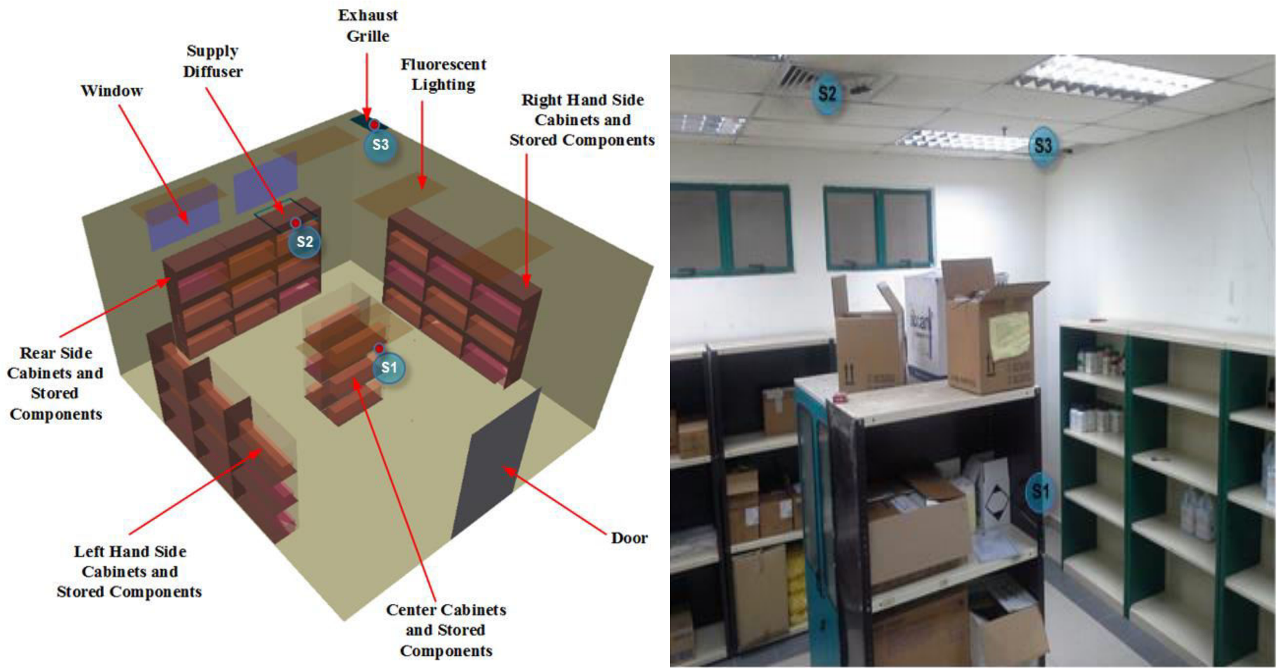
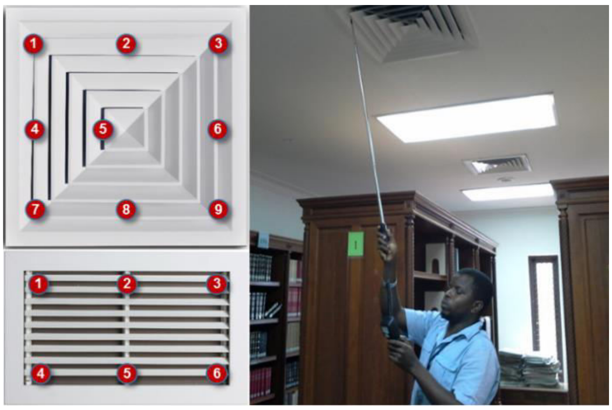


Figure-1. Space layouts with positions of supply, exhaust and other components of the case study room: Left - 3D model and right - field experiment. S1, S2 and S3 are virtual data loggers' positions.

The face of supply diffusers and exhaust grilles were divided into grids to facilitate measurement of airflow, thermal and hygric parameters of the outlets (supply and exhaust). Figure-2 shows the division of air outlets into grids with (a) supply diffuser (b) exhaust grilles and (c) field assessment of a typical supply diffuser with ALNOR AVM440 thermal anemometers. The measurements were averaged to give a fair representation of the measured parameters. The surface temperatures of the bounding walls, floor and ceiling were measured with Testo infra-red thermometer.Table-1 shows the specifications, accuracies and precisions of the measuring equipment.



(a) (b)

Figure-2. Division of air outlets into girds for better results accuracy (a) supply diffuser (b) exhaust grilles and (c) field measurement of a typical supply diffuser with ALNOR AVM440 anemometer.

Model setup

In this study 3D modelling (Figure-1) of the experimental room was created within the FLOVENT CFD analysis tool. The walls, floor and ceiling were model as adiabatic since the room is bounded with other rooms with similar air-condition. The furniture in the room comprise of metal shelves for keeping stored components. In the model, the shelves were placed nearly 25 mm from the wall surfaces and sat directly over the floor. The items were modelled as rectangular blocks to conserve computational resources. One of the many benefits of CFD simulation is the ability to place virtual data loggers at different points for replication of the measurement positions. The virtual data loggers allow assessment of the hygrothermal conditions at the different locations in the model.

CFD governing equations and boundary conditions

The fluid flow physics, heat transfer and other related processes are contained in the Navier-Stokes equations of the general form using the standard *k–ε* model as shown in Equation.(1)[11, 19].

$$\frac{\partial}{\partial t}(\rho\,\varphi)+div\left(\rho\,V\varphi-\Gamma_{\varphi}grad\varphi\right)=S_{\varphi} \qquad (1)$$

Where ρ is the density, V is the velocity vector, ϕ is the dependent variable in the flow field to which the equation applies (temperature, velocity, pressure, etc.), $\Gamma\phi$ is the turbulent diffusion coefficient and $S\phi$ is the source or sink term of the variable ϕ.

In CFD five of such equations are in use for the field variables: one equation for temperature (T) (derived

ARPN Journal of Engineering and Applied Sciences

from conservation of energy/enthalpy); three equations for x-velocity (u), y-velocity (v) and z-velocity (w) (derived from conservation of momentum) and one equation for pressure (p) (derived from the conservation of mass). Additional equations are coupled with these five set of equation for specie concentration (c), turbulence kinetic energy (k) and dissipation of turbulence kinetic energy (ε). In totality, the set of equations for coupled heat and mass transfer in CFD result in eight equations. The field variables (temperature, velocity, pressure) are function of x-coordinates, y-coordinates, z-coordinates and time. Table-2 shows the dependent variables (ϕ), effective diffusion coefficients (Γϕ), and the source term for each of the flow parameters.

In this study, the source terms are provided as boundary conditions from the air outlets - supply diffuser (temperature = 15.5 °C, humidity ratio = 9.8 x 10^{-3} kg/kg, airflow rates = 0.61m^3/s) and return grille (airflow rates = 0.61m^3/s). Others are enclosures surface temperatures – walls (17.4 °C, 16.8 °C, 18.4 °C and 19.6 °C), floor (18.4 °C) and ceiling (17.3 °C) and lighting heat gains (6 x 72 W). The return grille humidity ratio as well temperature are to be computed by the CFD simulation.

Table-2. The dependent variables (ϕ), effective diffusion coefficients (Γϕ), and the source term (SΦ) for each of the flow parameters.

Equation	Φ	Γ_Φ	S_Φ
Continuity	1	0	0
x momentum	u	μ	$-\partial P/\partial x$
y momentum (vertical)	v	μ	$-\partial P/\partial y$ - ρg
z momentum	w	μ	$-\partial P/\partial z$
Enthalpy	C_pT	λ	Q
Concentration	c/ρ	d	Q_m
k equation	k	μ/σ_k	$G - \rho\, \varepsilon$
ε equation	ε	μ/σ_ε	$C_1\, \varepsilon\, G/k - C_2\, \rho\, \varepsilon_2/k$

$\mu = \mu_{lam} + \mu_t$
$\mu_t = \rho\, C\mu\, k^2/\varepsilon$
$G = \mu[2[(\partial u/\partial x)^2 + (\partial v/\partial y)^2 + (\partial w/\partial z)^2] + (\partial u/\partial y + \partial v/\partial x)^2 + (\partial v/\partial z + \partial w/\partial y)^2 + (\partial u/\partial z + \partial w/\partial x)^2]$
C1 = 1.44, C2 = 1.92, Cμ = 0.09, σ_H= 0.9, σ_k= 1.0, σ_ε= 1.3

Discretisation and grid dependency analysis

The accuracy of CFD results depends largely on quality of the grids [11, 17]. The entire simulation domain was discretised using structured grid approach. Grid refinement was performed around the supply and return outlets to cater for high gradients often associated with air terminal devices thereby improving the prediction of velocity flow field [1]. The baseline grid was of coarse type that resulted in a total 4752 cells. The cell quantity was fine-tuned to medium and fine grids to give a total of 9885 and 39935 for medium and fine grids respectively. In this study, grid dependency analysis were carried out with virtual measurement of the hygrothermal parameters on a vertical section at center of the modelled room. Series of simulation were executed using coarse, medium and fine grids. Subsequent upon the satisfactory grid selection from grid dependency assessment, the model is further investigated for similarity between the measured and simulated parameters.

Model validation

In establishing the similarity between the measured and simulated parameters, the study adopted the percentage of root-mean square deviation (PRMSD) approach [4, 20]. PRMSD is obtained as a single value for all the points considered for both the measured and simulated parameters. It is evaluated from Equaiotn. (2) as follows:

$$RMSD = \sqrt{\left(\frac{\sum_0^n \left(\frac{Cm - C_S}{Cm} \times 100\right)^2}{n}\right)} \quad (2)$$

Where: C_m is the measured parameter; C_s is the simulated parameters and n is the number of points under considerations.

RESULTS AND DISCUSSIONS

Grid dependency analysis

Key parameters were measured for each of the grid experiments. The result is as presented in Figure-3. The prediction from coarse grid was far low compared to both medium and fine grids. Nevertheless, for most of the measured cases, the fine mesh under-predicts the basic flow parameters. This reason, coupled with longer computational time resulted in selecting the medium grid for further analysis.

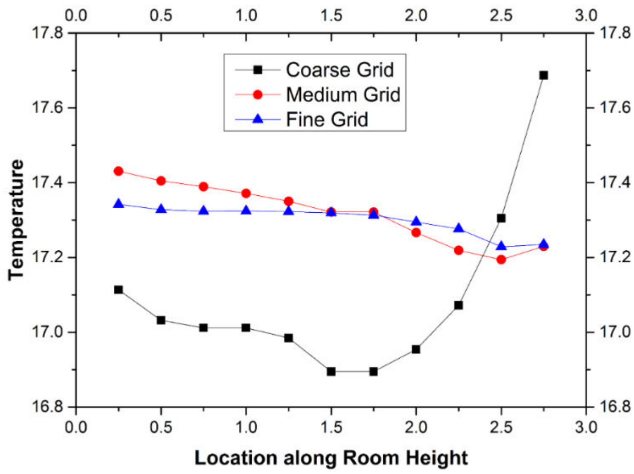


Figure-3. Grid dependency analysis.

Overall thermal and hygric profile

Figure-4a and 4b show the thermal and hygric profiles at center of the case studied room (x = 2.25m). A relationship is shown between the thermal and hygric profiles. In most cases, humidity becomes higher at lower

ARPN Journal of Engineering and Applied Sciences

temperatures and vice versa. The increase in humidity at low temperature is due to presence of more moisture in the air. At increased temperature, the moisture dries off leaving the air to be less humid and hence low relative humidity. The thermal profile shows thermal stratification in most part of the room. The upper part close to ceiling reveals some high temperature which is due to the heat emission from lighting. Cold region is concentrated on the air-stream close to the supply diffuser (Figure-4a). In addition, regions close to the shelves were found with low thermal gradients. Hence regions close to the shelves are found with higher hygric gradients (Figure-4b). Overall, the entire room still revealed thermal and hygric stratifications not only within the room but also around the shelves. These findings suggest that the air in the room is not well distributed. Consequence of such stratification in thermal and high hygric profiles could result in microbial proliferation as reported in a similar study elsewhere by the authors[21].

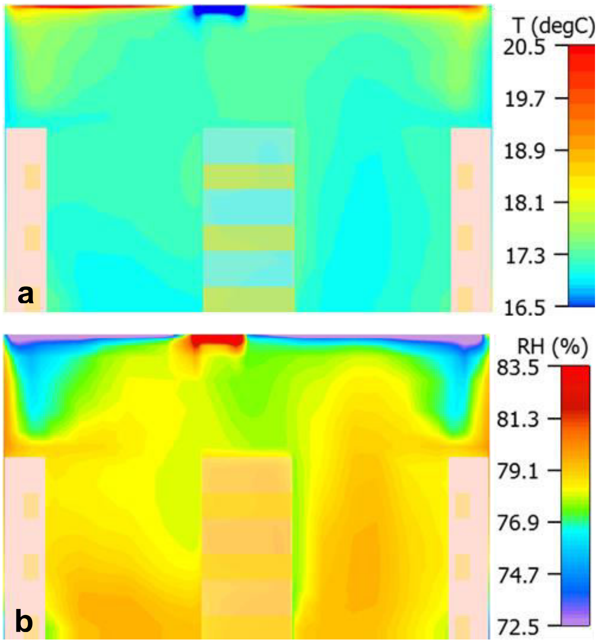


Figure-4. Ventilation profile in the case studied room with ceiling mounted supply and exhaust at x = 2.25m (a) Temperature contour (b) Relative humidity contour.

Validation of CFD simulation with measured data

Table-3 presents the results of in-situ and simulation experiments. The PMRSD between the measured and simulated parameters for the experiment results in 2.6% and 4.1% respectively for T and RH. These results follow submission from earlier studies that acceptable deviations between the measured and simulated values in numerical investigation should be in the range of 5% [22] and 10% [1]. In the study of [7], the difference was found to be higher than 5-10% where their experiment results in a deviation of more than 25%. A value of 6.7% deviation was reported in the work of [20].

Table-3. Comparison of hygrothermal parameters between measured and CFD simulations.

Location	Measured		Simulated	
	T (°C)	RH (%)	T (°C)	RH (%)
Supply Diffuser	15.5	85.1	15.8	83.5
Exhaust Grille	16.4	76.4	16.8	80.0
Room Ambient	17.9	82.4	17.3	78.4
PRMSD: T = 2.6% and RH = 4.1%				

The current work performs better than the earlier cited work. With recorded deviations of less than 5% in the current study, the hygrothermal parameters indicate that the CFD simulation produces a valid agreement with the measured parameters. Subsequent upon the satisfactory validation results, the numerical simulation can be used to further investigate the model for hygrothermal performance, microbial proliferations and other various indoor air quality problems.

CONCLUSIONS

The need for energy efficiency in building design, construction and operation has continuously received considerable attention within the past few decades due to high carbon footprint of the built environment. Although commendable improvements had been recorded on building energy efficiency, some detrimental effects had also been recorded as poor hygrothermal performance and associated deteriorative impacts. Computer simulation in the built environment is promising in the provision of fast and cheaper solutions to a host of challenges. Nevertheless, their use is inherently involves some degree of uncertainties. Therefore improving the certainty of simulation outcomes requires successive validation of the model. This paper reported the measurement and validation of a CFD model for the investigation of hygrothermal performance in an indoor environment with known cases of microbial proliferations. The methodology combines in-situ experiment with CFD simulation with a commercial tool using the standard k–ε model. Microclimate parameters obtained from in-situ experiments were used as boundary conditions in the CFD. Good correlation was established between the predicted and measured hygrothermal profile with less than 10% deviations. This indicates that the model can be employed for further investigation with high confidence.

ACKNOWLEDGEMENT

The results presented in this study is part of a multidisciplinary research project (FRGS12-067-0126) initiated on indoor mould growth prediction using thermal characteristics in buildings in the tropical climate by International Islamic University Malaysia and the Ministry of Higher Education (MOHE) Malaysia. The financial supports are gratefully acknowledged. The authors equally recognised the supports of the Dean of Kulliyyah (Faculty) of Medicine IIUM together with the management staffs of Daya Bersih during the studies.

REFERENCES

[1] Calautit J. K. and Hughes B. R. 2014. Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower. Energy and Buildings. 84: 367-377.

[2] Hajdukiewicz M., Geron M., and Keane M. M. 2013. Formal calibration methodology for CFD models of naturally ventilated indoor environments. Building and Environment. 59: 290-302.

[3] Wang Y., Fu-Yun Z., Kuckelkorn J., Liu D., Liu J., and Jun-Liang Z. 2014. Classroom energy efficiency and air environment with displacement natural ventilation in a passive public school building. Energy and Buildings. 70: 258-270.

[4] Haw L. C., Saadatian O., Sulaiman M. Y., Mat S., and Sopian K. 2012. Empirical study of a wind-induced natural ventilation tower under hot and humid climatic conditions. Energy and Buildings. 52: 28-38.

[5] Pérez-Lombard L., Ortiz J., and Pout C. 2008. A review on buildings energy consumption information. Energy and buildings. 40: 394-398.

[6] Wilson S. C., Palmatier R. N., Andriychuk L. A., Martin J. M., Jumper C. A., Holder H. W., *et al.* 2007. Mold contamination and air handling units. Journal of Occupational and Environmental Hygiene. 4: 483-491.

[7] Kavgic M., Mumovic D., Stevanovic Z., and Young A. 2008. Analysis of thermal comfort and indoor air quality in a mechanically ventilated theatre. Energy and Buildings. 40: 1334-1343.

[8] Chen Q., Lee K., Mazumdar S., Poussou S., Wang L., Wang M., *et al.* 2010. Ventilation performance prediction for buildings: model assessment. Building and Environment. 45: 295-303.

[9] Chen Q. 2009. Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment. 44: 848-858.

[10] Essah E. A., Sanders C. H., Baker P., and Kalagasidis A. S. 2009. Condensation and moisture transport in cold roofs: effects of roof underlay. Building Research & Information. 37: 117-128.

[11] Allard F., Awbi H. B., Davidson L., and Schälin A., Eds. 2007. Computational Fluid Dynamics in Ventilation Design (REHVA Guidebook. Finland: REHVA).

[12] Li Y. and Nielsen P. V. 2011. CFD and ventilation research. Indoor Air. 21: 442-453.

[13] Jelena S., 2011. Ventilation performance prediction: In building performance simulation for design and operation. J. Hensen and R. Lamberts, Eds., ed: Taylor & Francis.

[14] Majeed O. O. 2015. Mould Growth Prediction in Tropical Climate Buildings by Hygrothermal Differentials. Msc Dissertation, Building Services Engineering, International Islamic University Malaysia, Unpublished, 2015.

[15] Gaspar P. D., Barroca R. F., and Pitarma R. 2003. Performance evaluation of CFD codes in building energy and environmental analysis. Building Simulation 2003.

[16] Srebric J. and Chen Q. 2002. An example of verification, validation, and reporting of indoor environment CFD analyses (RP-1133). Ashrae Transactions. 108: 185-194.

[17] Chen Q. and Srebric J. 2002. A Procedure for Verification, Validation, and Reporting of Indoor Environment CFD Analyses. HVAC&R Research. 8: 201-216.

[18] Ashrae, Ashrae guideline 33-2013: guideline for documenting indoor airflow and contaminant transport modeling: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., 2013.

[19] Khalil E. E., Air distribution in buildings: CRC Press, 2013.

[20] Lim C. H., Saadatian O., Sopian K., Sulaiman M. Y., Mat S., Salleh E., and Ng K. C. 2013. Design configurations analysis of wind-induced natural ventilation tower in hot humid climate using computational fluid dynamics. International Journal of Low-Carbon Technologies. 0: 1-15.

[21] Ali M., Oladokun M. O., Osman S. B., Samsuddin N., Hamzah H. A., and Salleh M. N. 2014. Influence of indoor microclimate distribution on mould infestation in a university library. In In CIEC 2014: International

Civil Engineering and Infrastructure Engineering Conference, Kota Kinabalu, Sabah, Malaysia.

[22] Gong G., Xu C., Jiao J., Liu Y., and Xie S. 2011. Investigation of moisture condensation on papermaking plant envelopes in high humidity environment by orthogonal analysis and CFD simulation. Building and Environment. 46: 1639-1648.