Numerical Simulation Of Natural Ventilation For Full-Scale Office In Malaysia

Ali M. Wahhad, Abdulhafid M. Elfaghi

Abstract: The Natural ventilation is the sustainable solution to maintain the health and environmental conditions comfortable in buildings. This paper presents a 3D computational fluid dynamics (CFD) analysis to investigate indoor environmental conditions of naturally ventilated typical size of single office room of a high-glass wall fitted with two operable windows. The purpose of this work is to facilitate design and analysis of suitable window measurements. The numerical results obtained for a particular time (solar time) available and the maximum discrepancy obtained are 1.7% of temperature and 3% of velocity which are within the accepted level and computations with the CFD based model show best agreement with measurements and good thermal comfort is achieved when the outdoor temperature is moderate.

Index Terms: Window design; CFD, indoor environment; natural ventilation, Malaysian climate

1. Introduction

The Computational Fluid Dynamics (CFD) applications have achieved a great success in the field of natural ventilation. The effect of indoor airflow behavior in modifying the indoor thermal conditions in a courtyard residence found in the tropics. Airflow behaviors were sufficient to modify indoor thermal conditions to achieve comfortable environments. This correlation indicates that the potential of the courtyard to act as a passive cooling strategy is a function of the indoor airflow pattern [2]. Outdoors in exchange radiant conditions of the person body with the environment is particularly important due to exposure to the solar radiation, temperature. Other factors which affect the inside of the office are the movements of air and moisture vary much more in the outdoors than indoors. [3] have found that natural ventilation improves both the internal environment and worker productivity. Due to these benefits, a popular natural ventilation is coming and many have been encouraged research work in this area. Knowing of the air flow pattern and correct orientation of buildings, and location windows, doors and other openings is of paramount importance to provide good ventilation. Based on the available techniques to study the air flow and temperature are on experimental technique, and numerical and computational approach. CFD software is a powerful tool numerical simulation. However, the accuracy and reliability of the CFD predictions is still a major concern. The computational part of the study consisted of the steady-state application of three the RNG and k-ε models turbulence models. It was concluded that the numerical predictions obtained by turbulence models were generally in acceptable agreement with the experimental measurements. The RNG-k-ε model performed relatively better, especially for temperature predictions. This model is based on the model transport equations for turbulent kinetic energy, K, and its dissipation rate ε. [4] investigated the use of several turbulence models for predicting the velocity and temperature distributions for three flow situations in simple geometries.

- *Ali M. Wahhad, College of Engineering Technology Houn, Libya. E-mail: aliwhaad@yahoo.com*
- *Abdulhafid M. Elfaghi, Aeronautical Engineering Department, Universiti of Zawia, Libya, PH-00218913671983. E-mail: hafied@zu.edu.ly*

Kobayashi et al (2009) applied standard k-3 model and model of Reynold stress for the study of transfer the power and loss for cross natural ventilation in the pitched ceiling and buildings of rectangular shape. Tominaga et al (2008). and Ramponi and Blocken (2012) discuss application CFD models to simulate the wind around buildings and found that the domain of computational, the size of the mesh, boundary condition, the scheme numerical and convergence affects the results of the simulation, and have concluded that the setup of numerical should be checked carefully. The numerical predictions were compared with the experimental measurements. It was concluded by the authors that all the turbulence models gave satisfactory results and the choice of model used depends on time and computational power restraints. James, O.P. et al (2011) examined with effects of hypothetical apartments in a building on natural ventilation by solving the cross-ventilation rate, and demonstrated the exercises to validate model agreements between convincing results CFD and experimental measurements. [5] Studied natural cross-ventilation in a test room with openings at non-symmetrical locations experimentally and numerically. [6] have evaluated the laminar, the standard k-ε and the RNG k-ε models with respect to their performance in simulating the flow in a model room. Their simulations have found the RNG k-ε models agreed better with experimental data than calculations with the standard k-ε model. Many investigations have found numerical simulation to be a valuable tool of room ventilation design [7-14] however the increasing computational power of computers, reliance on an internal simulation of the air flow prediction and increase usage. Most of these comparative studies conclude that k-ε models are capable to simulate indoor airflows but the RNG k-ε model is slightly better than the other models in terms of the overall simulation performance. The objective of the present study is to validate the air flow pattern and temperature distribution in a room using CFD analysis. Some studies deal with the complex situations but did not consider the interactions of solar radiation with convective air flows. This study was undertaken in order to evaluate K-ε turbulence model in combination with a radiation model for the prediction of air flow and temperature distributions inside the office of geometrical configuration in building under neutral ventilation conditions for which measurements data are available.

2. DESCRIPTION OF THE SELECTED BUILDING

The field measurement room was conducted at Faculty of Engineering, Universiti Putra Malaysia. The building was built in 2005, consists of seven-storey and is geometrically facing and orientated at 22 degrees from true north to the east. The shape of the building is cuboid model as shown in Figure 1. The office dimension (y-direction); length (x-direction) and height (z-direction) was 2.85 m 3.80m 2.60 m respectively. The office model has demonstrated the use of large glass areas dimension 2.30m, 2.59 m. Two open glass windows (6mm) can be opened with angle 30^o. Thin silver film material was coating glass window to reduce heat from solar radiation. The height and width of windows were 1.14 cm x 52 cm.

Fig. 1 *Faculty of Engineering, University Putra Malaysia*

3. CFD MODEL

In the present CFD modeling work, it was assumed that the air flow is steady, turbulent and three-dimensional; the time-averaged governing equations can be used in conjunction with a turbulence model to adequately predict the airflow. The air-flow is single phase, the effects of dust particles and water vapors are neglected, the airflow at any inlet vent is uniform, the air properties are constant, except for the density change with temperature that gives rise to the buoyancy forces, this being treated using the Boussinesq Approximation, and external ambient conditions are steady. Turbulence model were tested in this study renormalization group k-epsilon (RNG k-ε) model for the assessment of indoor thermal environment. In obtaining the numerical solution, the computational mesh was generated using the program GAMBIT and governing equations were numerically solved using the commercial computational fluid dynamics ANSYS CFD, solver FLUENT. The SIMPLE algorithm was adopted to couple the pressure and momentum equations. When the sum of the absolute normalized residuals for all the cells in flow domain became less than 10^{-6} for energy and 10^{-3} for all other variables, the solution was considered as converged. Grid dependence of each case was tested using three grids to ensure that grid resolution would not have a notable impact on the results. Imitate settings from CFD to model indoor during field measurements. For validation, the numerical results were compared with the available experimental results.

4. METHODOLOGY

This paper investigates airflow and difference temperature between indoor and outdoor environmental conditions of a naturally ventilated office room. This investigation was facilitated by the solar-time field measurements in the office and the formal methodology for reliable and robust CFD models of indoor environments. The verification and validation of CFD model, and a parametric analysis to evaluate the impact of boundary conditions on the results of the simulation model are presented. Results of the calibration procedure for CFD model that, based on field measurements, accurately represents are in indoor environment. Where verified the grid takes place. Various runs of the initial CFD model were performed on different size meshes and the results are compared to the independence of the grid analysis solution. Independent grid solution implies the results do not change significantly with increasing the number of grid cells, which is to achieve a balance between accuracy and computational time. In this present work, the solution is evaluated quantitatively independent grid. Once the grid independence is established, the simulated air velocity and air temperatures inside the office are validated with on-site measurements. The last step is the process of improving the agreement between the measurements and simulation data by adjusting the most relevant input parameters. This step should be repeated as long as the CFD model verification meets the criteria of being a good representation of the real environment. The geometry used in the computations and is approximately the same with the original geometry as shown in Figure 3. Boundary conditions are defined in the main components of the model room. Boundary conditions were set equal to the experimental data. The inlet values of the air velocities are set equal to the experimental. The wall surfaces were assumed to be adiabatic. The average temperature is determined to represent the actual temperatures room. The radiation model is used. Heat temperatures are modeled to represent the actual amount of heat sources of the room. Heat emitted by the computer is set equal to 300K and inlet temperature is set to 303.15K. The distributions of k-ε are assumed to be uniform. The mix thermal boundary conditions were used for the facade glass surface. The modeling of the glazing was simplified as a single glazing and effective thermal conductivity of 0.0626 W/m2K for the glazing with a thickness 6 mm was used. Average outdoor static pressure is equal to atmospheric pressure gradients are set from all other variables equal to Unstructured hybrid element mesh generator with an adaptive mesh refinement algorithm automatically is employed, which permits a very accurate representation of the boundaries. The advantage of using unstructured mesh is to reduce the numerical errors and consistency of cross-domain solution. The numerical grid consists of approximately 39030 unstructured elements, and hexahedron type of mesh has been used for the six shapes window, office, corridor, table, computer and human as shown in Figure 4.

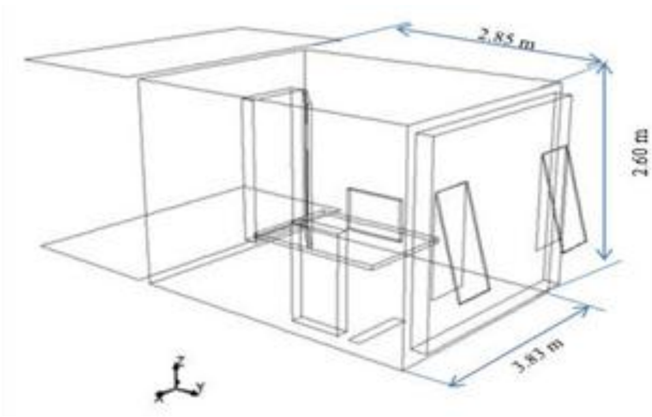

Fig. 3. *Geometry of CFD model*

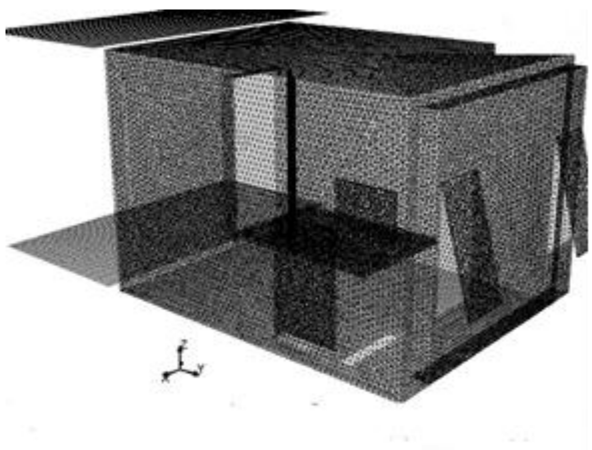

Fig. 4. *Mesh model*

5. Results and Discussion

The results were used to determine the average air temperature, temperature difference and airflow rate that can be collected in the office with the variables of different solar time and validation experimental work by CFD Fluent. The distributions of air temperature in the office is shown in Figure 5. The calculated flow field using the CFD k–ε model is shown in the central yz plane (at x = 3.83 m), and in a horizontal x z plane (at y = 2.85). The temperature on the front windows was high due to absorption and emissivity heat through the glass into the office with airflow to ranging from 300K to 309K. It can be seen the temperature in floor was high 4K than ceiling due to radiation through windows.

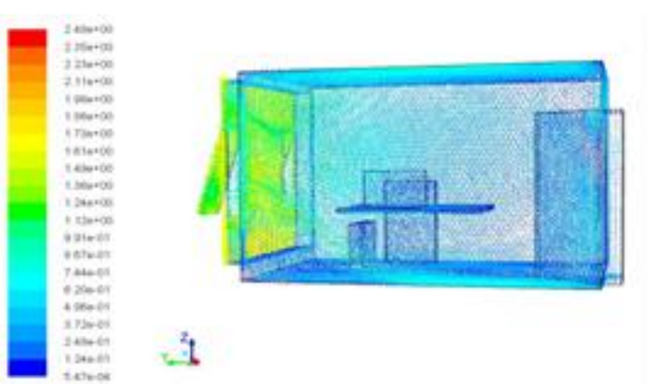

Fig. 5. *Distribution of velocity in the office at 8:00 am*

Figure 6. Show the airflow and layers of natural ventilation inside the office and through the windows. The air flow inside the office, and bends downwards due to buoyancy and forms a ground and ceiling layers, which flows along the floor and ceiling with velocities of ranging from 0.05 to 0.13 m/s. The velocity in the office is ranging from about 0.11 to 0.22m/s in layer 1 m from window and less than 0.1m/s in layer 0.5 m from window. It can be seen in each case the velocity natural ventilation near window zone is higher than the center zone of office, because outdoor air was supplied from window openings, which ranging from 0.61 to 1.13 m/s.

Fig.6. *Distribution air temperature in the office at 8:00 am*

The computational results were compared with the results obtained from thermal analysis. There was a very good agreement between the results of measurements and CFD. Figure 7 represents the average hourly temperatures (measured and simulated) in an office. Average deviations between the temperatures (measured and simulated) are below 1oC. The biggest differences appear in 13:20 when solar radiation influences. This is due to multiple solar radiation reflections through the air gap and to the temperature rise on the window's surface. Despite this fact, the comparison of results proves that the simulation agrees quite well with experimental measurements.

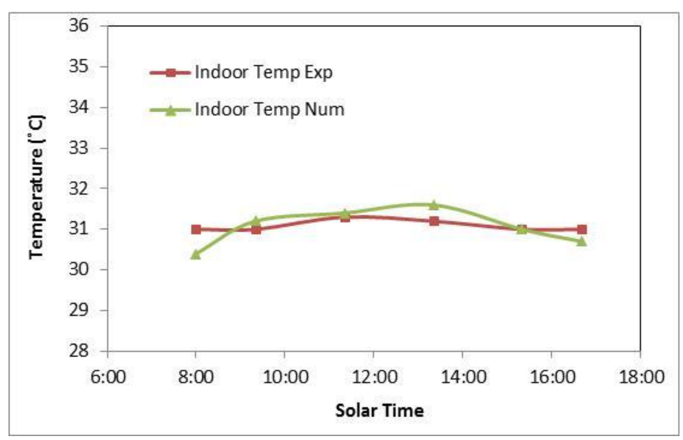

Fig. 7. *Averaged Temperature comparison between measurement data and CFD*

Predicted results are compared with those from experiments. Figure 8 shows a comparison between the airflow pattern predicted by CFD and the airflow rate values obtained by measurements and comparing the results with CFD. These simulations were carried out for stack effect only and wind effect. In these simulations the indoor temperature was 30oC (303K).

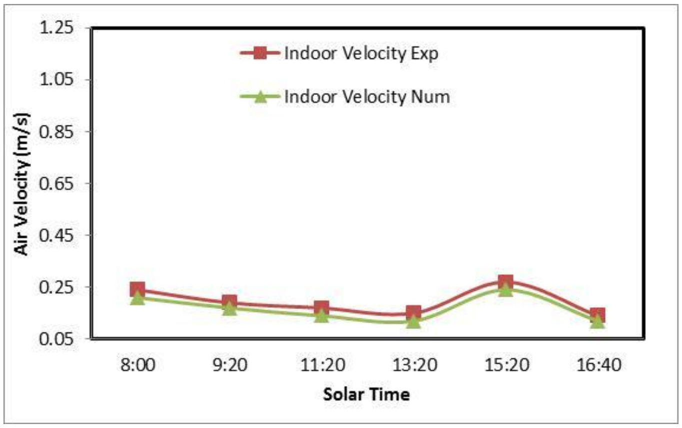

Fig. 8. *Air velocity comparison between measurement data and CFD*

6. Conclusion

This paper presents a 3D computational fluid dynamics (CFD) analysis to investigate indoor environmental conditions of naturally ventilated typical size of single office room of a highs-glass fitted with two operable windows. CFD technique was used to simulate the internal air flow pattern and temperature. The CFD simulated result was checked for grid independency and standard k-ε model is selected as a suitable turbulence model. The CFD simulation airflow and temperature of the office were compared with the experimental results for its validation. Also the CFD simulation airflow pattern and temperature distribution inside the office were compared with the experimental results for its validation. The maximum discrepancy obtained between the computed CFD of air temperatures, velocity are 1.70% of temperature and 3% of velocity respectively. The difference between the computed CFD of air temperatures and the measured data is generally less than 1K and velocity range 0.24m/s. From this paper, the most un-comfort zone was identified nearer to the wall window and the maximum discrepancy obtained are1.70% of temperature and 3% of velocity.

ACKNOWLEDGEMENTS

The authors would like to thank Mechanical and Manufacturing Engineering Department, University Putra Malaysia.

REFERENCES

[1] S.K Chaturvedi, T.O Mohieklin, A CFD anaylisis of effects of vent-location on pollutant concentration in rooms, in: Proceedings of the conference on Building Systems: Room Air and Air Contaminant Distribution ASHRAE, University of Illinois press. Urbana-Champaign, II, USA, 1988, pp. 158-160.

[2] Rajapaksha I. Passive Cooling in the Tropics: A Design Proposition for Natural Ventilation, The 21st Conference on Passive and Low Energy Architecture. Eindhoven, the Netherlands, 19 – 22 September 2004. University of Moratuwa, Moratuwa, Sri Lanka

[3] Guohui Gan, (2006). Simulation of buoyancy-induced flow in open cavities for natural ventilation, Energy and Buildings 38 (5) 410–420

[4] Fisk W.J and Rosen A.H. (1997) Estimates of improved productivity and health from better indoor environments Indoor Air. 7: 158-172.

[5] M. Cable, P.H. Oosthuizen, An evaluation of turbulent models for the numerical study of mixed and natural forced convective flow in atria, in: Proceedings of the 2nd Canadian Solar Buildings Conference, Calgary, June 10–14,2007.

[6] G. Stavrakakis, M. Koukou, (2008) Natural cross-ventilation in buildings: building scale experiments, numerical simulation and thermal comfort evaluation, Energy and Buildings 40 (9) 1666–1681

[7] Tahir Ayata, (2005) Temperature distributions in a copper and aluminum layered base of a CrNi-steel saucepan Original Research Article Applied Energy, Volume 80, (4), Pages 341-347

[8] Posner JD, Buchanan CR, Dunn-Rankin D. (2003) Measurement and prediction of indoor air flow in a model room. Energy and Building; 35(5):515–26.

[9] Busnaina A.A., Salem A., Zharif M.A.R., Three-dimensional numerical simulation of fluid flow and particle transport in clean room in: proceedings of the conference of institute of environmental sciences, Institute of environmental Sciences, 1988,pp.326-330.

[10] Chen Q. Comparison of different K-ε models for indoor air flow computations, Numerical Heat transfer, Part B 28 (1995)353-369.

[11] Choi H.L., Albright L.D Timmons M.B., An application of the K-3 turbulence model to predict how a rectangular obstacle in a slot-ventilated enclosure affects air flow, transactions in Agriculture 33(1) (1990)274-281.

[12] Chow W.K., Fung W.Y. (1996) Numerical studies on the indoor air flow in the occupied zone of ventilated and air conditioned space, Building and environment 31(4) 319-344.

[13] Fan Y. (1995) CFD modeling of air flow and contaminant distribution in rooms, Energy and Buildings 23(1) 33-39.

[14] Jones P., Waters R, The practical application of indoor air flow modeling, in N. Nagada (Ed.) proceedings of the conference on modeling of indoor air Quality and Exposure, American Society for Testing and Materials, Philadelphia 1993, pp. 173-181.