Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

CFD Simulation on Ventilation of an Indoor Atrium Space

Open Access

Nor Azirah Mohd Fohimi[1], Muhammad Hanif Asror[2,*], Rosniza Rabilah[1], Mohd Mahadzir Mohammud[1], Mohd Fauzi Ismail[1], Farid Nasir Ani[2]

[1] Faculty of Mechanical Engineering, Universiti Teknologi MARA, Cawangan Pulau Pinang, 13500 Permatang Pauh, Pulau Pinang, Malaysia
[2] Fakulti Kejuruteraan Mekanikal, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

ARTICLE INFO

Article history:
Received 18 March 2020
Received in revised form 17 May 2020
Accepted 22 May 2020
Available online 30 May 2020

Keywords:
CFD Simulation; Hybrid Ventilation; Atrium Space

ABSTRACT

Atrium is a large enclosed space with various shape and height. The design of the atrium is exceptional and unique which gives a challenge and complexity to architect and engineer to provide acceptable thermal. In this study, simulation ventilation of an indoor atrium space was analysed. A previous study indicates that CFD simulation can be used to successfully simulate the heat transfer and fluid flow in atrium geometries and provides recommendations turbulence and radiative heat transfer modelling. The variables which affect the atrium ventilation system can be categories into 3 elements which are external variables such as ambient and outdoor temperature, solar radiation and wind, secondly, an internal variable such as internal heat load, expected comfort level and the last one ventilation technique such as natural ventilation or forced ventilation. The purpose of this study is to predict the air ventilation system and evaluate thermal comfort in an atrium using CFD simulation. An atrium of an indoor theme park will be used as the case study. Since the location of the atrium is located at a hot-humid climate, the thermal comfort range is higher than expected by international standard, ASHRAE 55 2004 Standard. However, the standard still can be used to validate the result of the simulation by comparing it with Malaysia Standard 1525:2007. It was found that CFD simulation can predict the air ventilation and evaluate thermal comfort with less than 3% of percentage error between site measurement and simulation result.

1. Introduction

Atrium is a large enclosed space attached to the building with various shapes and typically has significant height. As the design of the atrium is very exceptional and unique, it gives a challenge and complexity in providing thermal comfort especially in terms of energy consumptions. One of the requirements for atrium design is to satisfy thermal comfort design following the international standard.

* *Corresponding author.*
E-mail address: hanif.ijm.uitm@gmail.com (Muhammad Hanif Asror)

https://doi.org/10.37934/cfdl.12.5.5259

The building energy consumption can be reduced by properly designed ventilation system of the atrium. A poorly designed atrium will cause unachievable thermal comfort inside the atrium and the worst will increase the energy utilized to cool the building. A low energy cooling technology may be considered as a passive and hybrid cooling system [1]. A hybrid ventilation system can be described as a two-mode system using a buoyancy-driven and mechanical cooling system or combination of both at different times of the day [2].

Natural ventilation is a practical way to reduce energy consumption but will risk thermal comfort. De Dear and Brager [3] state thermal comfort range for building ventilated by natural ventilation has a wider acceptance range compare with standard mechanical HVAC system.

According to Chan, Riffat and Zhu [4] the natural ventilation capable to reduce 60% of the total building consumption. However, the natural ventilation required profound knowledge and accurate prediction of the airflow and heat transfer to quantify naturally driven ventilation rates [5]. A limited number of CFD studies have been carried out on atrium buildings provided experimental validation for their CFD models [6]. CFD has been used to simulate the performance of natural ventilation to the ventilation system [7]. CFD is being increasingly employed to predict the building airflows and testing the ventilation strategies (e.g. in Ref. [8-10]).

Rundle [6] indicates nature physical phenomena govern fluid flow & heat transfer within atrium can be categorized by turbulent natural convection, radiative heat transfer, and conjugate heat transfer. Study has found the air-flow intensity becomes even higher when a combination of ventilation incorporated [11]. As the combination of all this phenomenon, it results in a complex calculation that required computational fluid dynamics (CFD) to simulate the heat transfer and fluid flow in complex atrium geometry. The application of CFD also can be used to evaluate the thermal comfort conditions of the atrium.

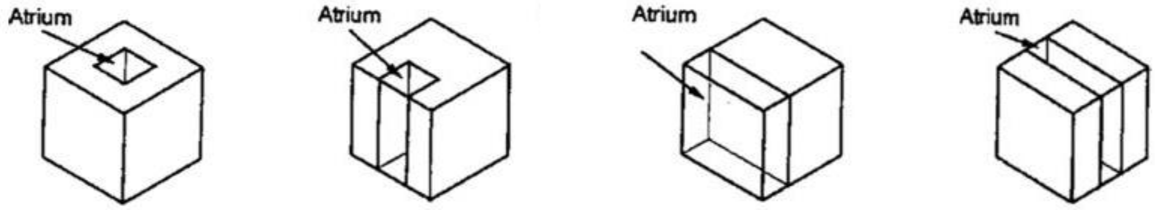
Even there are some research regarding simulation of air ventilation, but few studies have examined atrium as the case study. As irregular geometry of atrium, it is significant to investigate the suitable ventilation to achieve thermal comfort level. The aim of this paper is to predict air ventilation of an indoor atrium space using CFD simulation and the evaluate thermal comfort of the atrium. For the CFD results, the simulation program Ansys Fluent is used. The influence on the results of computational mesh and the thermal boundary conditions is considered. Most importantly, it is illustrated that the agreement of simulation results and experimental data is demonstrated for a wide range of tests.

2. Methodology

2.1 Building Description

Atrium nowadays often be designed with a glass wall façade and aluminum roof creating a common space interconnecting the adjacent floor and stories. The design of the atrium is generally based on the functionality and the architect’s aesthetic value. The position of the atrium to the adjacent of the building will be the main factor to gain the efficiency of the ventilation system with the expected thermal comfort. There are four types of the atrium as shown in Figure 1 below.

The atrium of an indoor theme park for the case study is in a hot and humid area. The total height of the atrium is about 29.5 meter, consists of 4 occupied level. The atrium is fully ventilated by forced ventilation from the fan coil unit located at the ground floor of the atrium and has outlet louvers at level 4. The exterior of the atrium is covered by a glazed façade.



(a) (b) (c) (d)

Fig. 1. Atrium types: (a) centralized, (b) semi-closed, (c) attached and (d) linear

2.2 Mesh Model

For grids generation, the main consideration is the accuracy of numerical solver in modeling the airflow profile. The numerical solver would perform the iterations based on numerical models employed for the calculation from one cell to the adjacent cells. In the selection of a grid or a mesh type for the model, the setup time issue is the most considerable item. The creation of mesh using triangular and a tetrahedral element resulted in fewer cells as compared to the equivalent mesh consisting of quadrilateral or hexahedral elements. Hence, it enhanced the setup time and reduced computational time of iteration until an acceptable convergence was attained. A fine mesh is used with mesh smoothing sizing characteristic based on tetrahedron assembly method which results in 34449 nodes and 175423 elements Figure 2.

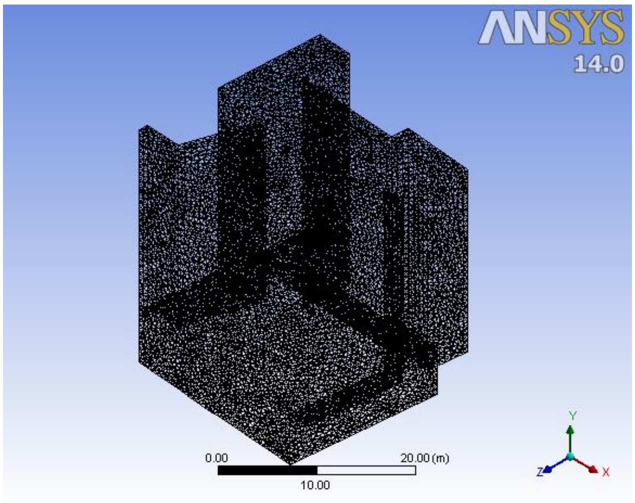


Fig. 2. Mesh modeling of the atrium

2.3 Boundary Condition

To determine the boundary conditions which cannot be measured at the investigated area, and information from literature review been used such as the inlet air flow of the cassette unit, room/wall temperature, inlet air temperature, and outlet louver pressure. These boundary conditions are been used as the boundary conditions which been setup in CFD simulations. The details of boundary conditions are tabulated in Table 1.

Table 1
The boundary conditions in CFD simulation

Inlet Air Flow Velocity	5 m/s
Inlet Temperature	18 °C
Outside temperature	27 °C

2.4 CFD Simulation Procedure

Using commercial CFD software ANSYS Fluent, the CFD turbulence model is Reynolds Averaged Navier-Stokes (RANS) modeling approach with SST k-ω model as the turbulence model and the Discrete Transfer Radiation Model (DTRM) as the radiation model. There are several assumptions made (i) the flow is turbulent and three dimensional, (ii) the flow is single phase with dust particles and water vapour are neglected, (iii) the flow at an inlet vent is uniform, (iv) the air properties is constant except for the density of air changes with temperature of buoyancy driven forces which used Boussinesq approach and (v) external ambient conditions are steady.

The CFD software solves the equations for conservation of mass, momentum and total energy. For spatial discretization, the pressure-velocity coupling scheme used is SIMPLE while for pressure scheme is Body Average Weighted, and the momentum scheme is 2nd Order Upwind Scheme. To achieve a converged solution, the iteration will be based on the energy equation until residual fell below 10^{-6} and absolute normalized residual less than 10^{-4}.

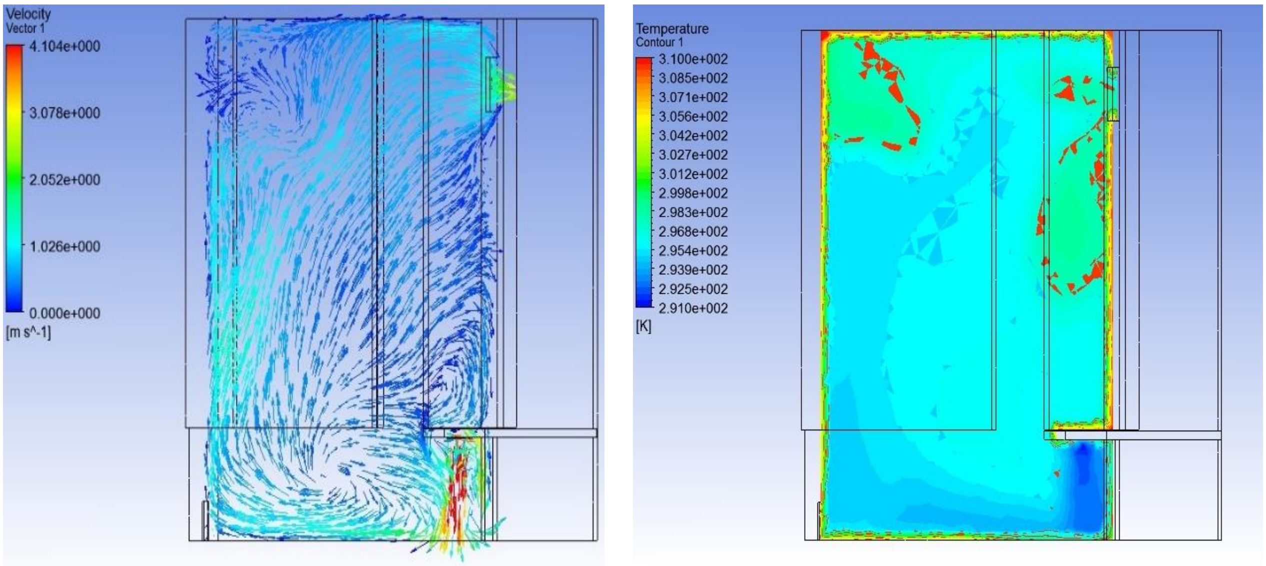
3. Results

3.1 CFD Simulation Result

The atrium is ventilated by mechanical ventilation located at the ground floor high level. 2nos fan coil unit with 9000Btu/h and 250 CFM ducted to 3nos linear diffuser is used to cool the atrium with average air velocity per diffuser 5 m/s.

Figure 3(a) shows the velocity vector along the middle vertical planes to the façade of the atrium. This plane is selected for analyzing the effect of the glazed façade on the velocity and temperature. The highest velocity located at the inlet of the fan coil unit as in the simulation, it is set as the inlet velocity to the atrium with an average of more than 4 m/s. The air then flows near the ground slab towards the façade than thru outlet louvers. From the velocity vector, there is a certain area where the air is not flow which is at the level 1 slab and the top corner of the atrium. From the velocity vector, it clearly shows the air flow is not fully distributed to the whole space of the atrium. The maximum air flow located at ground floor level while there almost no air flow at level 1. The air flow focus on the outlet louvers as it is the only outlet located in the atrium. The air flow also minimizes at the top corner of the atrium as it is the furthest location from the fan coil unit. The air flow at the top corner and middle of the ground floor experience turbulence as the air of this area moves in the opposite direction.

Figure 3(b) shows the temperature distribution along YZ-axis with at the middle of X-axis. The atrium is fully ventilated by the fan coil unit and the inlet temperature is set by 18°C (291.15 K) as the minimum achievable inlet temperature of the fan coil unit. The outlet louvers are set as the average atmosphere temperature surrounds the atrium which is 27°C (300.15 K). Based on the temperature distribution the lowest temperature is located below the fan coil unit which set as the inlet fan. The temperature gradually increases along the ground floor slab towards the façade. The temperature shows significant increasing along the façade vertically. This is because of solar radiation transmit along the façade glazed wall. At the top corner of the atrium, the temperature already exceeds 300K as the combination effect of the solar heat radiation and insufficient air flow. The temperature also exceeds 300K underneath the outlet louvers but slightly reduce to 296K near level 1 floor.



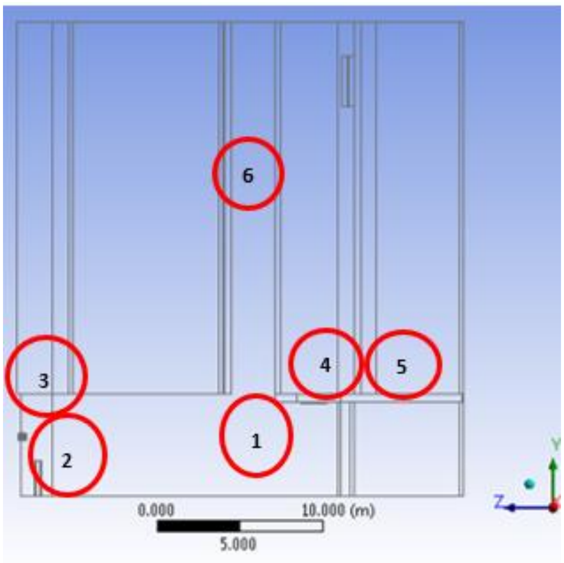
(a) (b)

Fig. 3. (a) Velocity vector along the middle vertical planes parallel to the façade of the atrium and (b) Temperature distribution along Y-Z axis with at the middle of X-axis

From this temperature distribution the thermal comfort at two occupied area (ground floor and level 1) is still achievable as the average temperature at ground floor is 294.5K (21.35°C) while at level 1 is 296.8K (23.95°C). Previous research suggested the thermal comfort in hot and humid climate is slightly higher than the international standard. The dry bulb range is suggested that between 23°C to 28°C. Hence, the inlet temperature can be slightly higher to reduce energy consumption to cool the atrium.

3.2 Site Measurement for CFD Validation

The data from result CFD simulation is validated by comparing it with site measurement data. The site measurement is focused on 6 numbers of localized temperature and air flow pressure difference located inside the atrium as shown in Figure 4. Two equipment is used to conduct site measurement 1) sling psychrometer and 2) anemometer + manometer. The comparison table is shown in Table 2.



Location 1 = GF Center of Atrium
Location 2 = GF Main Entrance Door
Location 3 = 1F Near Façade Glass
Location 4 = 1F Ahu Door Close
Location 5 = 1F Ahu Door Open
Location 6 = 4F Outlet Louvers

Fig. 4. Six Location for Site Measurement and CFD comparison

Table 2
Table of temperature between site measurement and CFD simulation with percentages differences

Measurement Device	Temp (°C)					
	Loc 1	Loc 2	Loc 3	Loc 4	Loc 5	Loc 6
Sling	26.0	27.5	28.0	23.0	23.0	31.5
Anemometer+ Manometer	25.73	27.30	27.90	23.03	23.35	28.30
CFD Simulation	25.92	27.55	27.46	23.01	23.01	30.74
Percentages Differences	0.21%	0.54%	1.78%	0.02%	0.73%	2.73%

3.3 Buoyancy Driven Natural Ventilation Effect

After the CFD Simulation result have been validated, the effect of natural ventilation system can be simulated. Natural ventilation in an atrium mostly occurs through the buoyancy-driven force [12]. Buoyancy driven natural ventilation occurs when the indoor temperature is greater than the outdoor temperature. These temperature differences develop pressure differences due to indoor air density is lower than outdoor air which causes air movement throughout the building [13]. A buoyancy driven natural ventilation is been introduced in the simulation by adding a free flow louver at GF high level. In this paper, conduction and radiation effects are neglected to meet the assumptions of the mathematical models and the experiments by Joanne *et al.,* [14].

The temperature inside the atrium show greatly increases compared with the current ventilation system and exceed thermal comfort. The average temperature at the middle of GF atrium is 27.64°C which exceeds ASHRAE 55 standard requirement to not exceed 26°C as shown in Figure 5(a). It indicates that the temperature of almost all the occupied areas of the atrium lobby ground floor is within the comfort range at the recorded time, whereas the bridges exceeded the comfort limits [15].

The buoyancy driven natural ventilation will not happen to the atrium unless a frequent strong wind blows toward the atrium and push the cool air from leaving the atrium through the louvers. The air movement inside the atrium also shows the air is not fully distributed along the atrium. As the cool air is moving towards the louvers at GF high level, the cool air cannot reach a higher level of the atrium. The air from higher levels also shows movement towards the GF louvers and reducing the amount of air moving outside from L4 louvers compared to the previous simulation as shown in Figure 5(b).

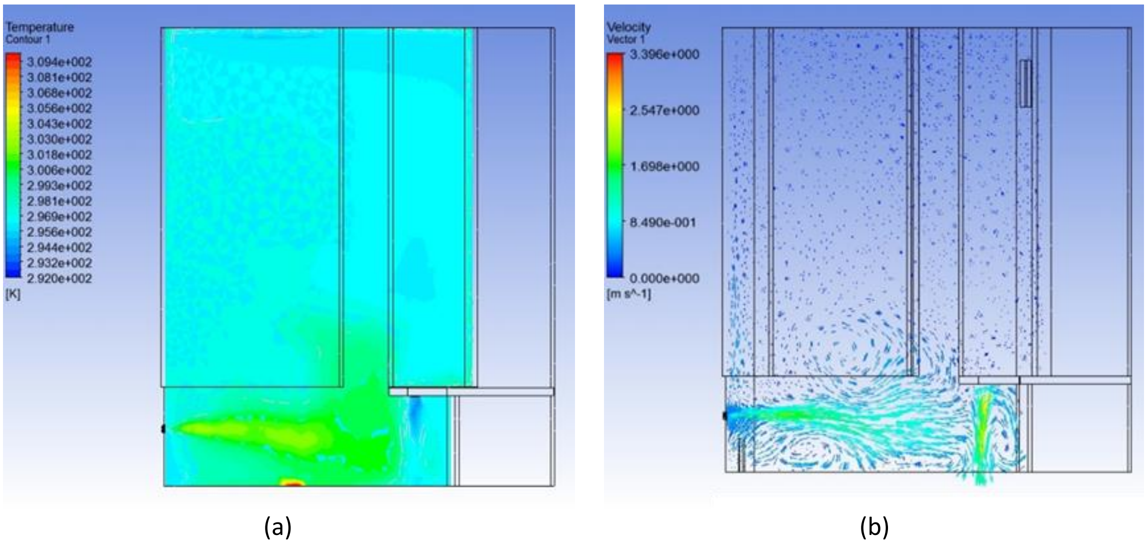
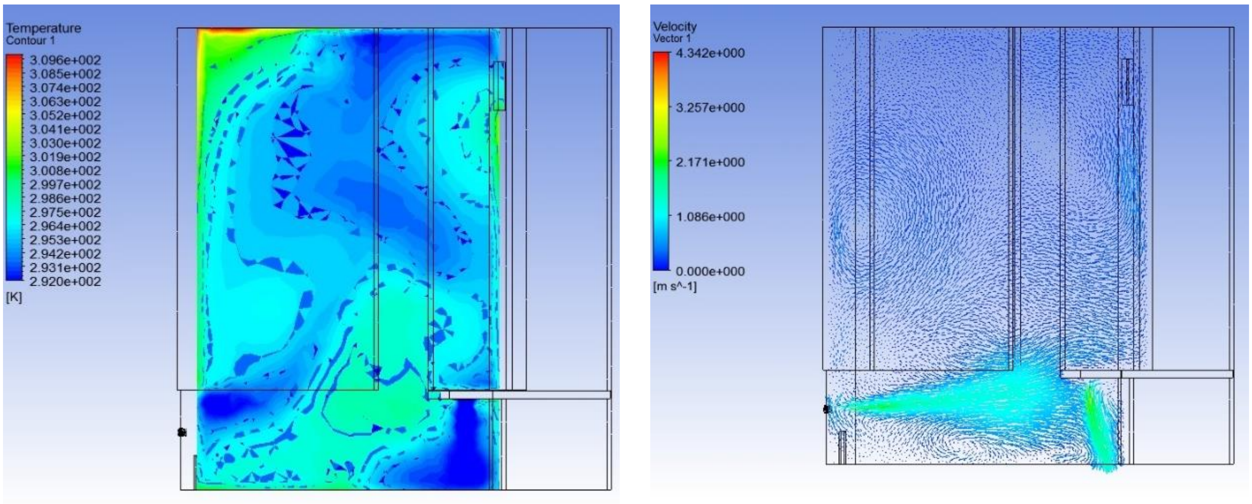


Fig. 5. (a) Temperature distribution along Y-Z axis with at the middle of X-axis with the effect of buoyancy driven natural ventilation and (b) Velocity vector along the middle vertical planes parallel to the façade of the atrium with the effect of buoyancy driven natural ventilation

3.4 Hybrid Ventilation Effect

As the buoyancy driven natural ventilation is not acceptable to satisfy the thermal comfort of the atrium, a hybrid ventilation system can be introduced. Hybrid ventilation is a combination of stack driven natural ventilation and forced ventilation. An inlet fan is introduced in the simulation while still maintaining the FCU inlet at the GF level. The simulation used 27°C as the inlet fan temperature with 7 Pa as the inlet pressure through the fan.

The result of CFD by using hybrid ventilation show improvement of temperature inside the atrium compared to simulation by using buoyancy driven natural ventilation. The temperature at GF is around 25.76°C which is slightly higher than the current temperature which used full forced mechanical ventilation. Although the temperature is higher compared to full forced ventilation, it is still in the acceptable region of thermal comfort (23°C ~ 26°C) as shown in Figure 6(a). As the hybrid ventilation is using inlet fan the air movement inside the atrium is moving in from outside. This movement is better compared to the case using buoyancy driven louvers as the cool air produced by forced mechanical ventilation is not leak to outside but be separated inside the atrium. The air moves along the atrium and moves toward the L4 louvers same as the case with the current ventilation system as shown in Figure 6(b).



(a) (b)

Fig. 6. (a) Temperature distribution along Y-Z axis with at the middle of X-axis with the effect of hybrid ventilation and (b) Velocity vector along the middle vertical planes parallel to the façade of the atrium with the effect of hybrid ventilation

4. Conclusions

This paper has successfully demonstrated CFD Simulation of Ventilation System of indoor theme park atrium. The numerical results obtained indicates that the RANS turbulence model includes in SST k-ω turbulence model concurrent with DTRM model can predict the ventilation system in the atrium. A buoyancy driven natural ventilation is not suitable to be applied as the indoor temperature will increase because of outside temperature and air movement towards the outside. A hybrid ventilation between forced mechanical the ventilation and forced natural ventilation can improve air distribution while maintaining thermal comfort inside the atrium.

Acknowledgement
The authors would like to thank En. Nur Izdihar Mohd Sidek as granted us permission to conduct site measurement inside the atrium theme park at Puteri Harbour, Johor. Gratitude is also expressed to Dr. Haslinda Mohamed Kamar and the FKM UTM Thermodynamic Laboratory Staff for allow us to borrow the measurement device. This research was not funded by any grant.

References

[1] Hussain, Shafqat, and Patrick H. Oosthuizen. "Numerical investigations of buoyancy-driven natural ventilation in a simple atrium building and its effect on the thermal comfort conditions." *Applied Thermal Engineering* 40 (2012): 358-372. https://doi.org/10.1016/j.applthermaleng.2012.02.025
[2] Hussain, Shafqat, and Patrick H. Oosthuizen. "Validation of numerical modeling of conditions in an atrium space with a hybrid ventilation system." *Building and Environment* 52 (2012): 152-161. https://doi.org/10.1016/j.buildenv.2011.12.016
[3] De Dear, Richard J., and Gail S. Brager. "Thermal comfort in naturally ventilated buildings: revisions to ASHRAE Standard 55." *Energy and buildings* 34, no. 6 (2002): 549-561. https://doi.org/10.1016/S0378-7788(02)00005-1.
[4] Chan, Hoy-Yen, Saffa B. Riffat, and Jie Zhu. "Review of passive solar heating and cooling technologies." *Renewable and Sustainable Energy Reviews* 14, no. 2 (2010): 781-789. https://doi.org/10.1016/J.RSER.2009.10.030.
[5] Zhai, Zhiqiang John, Mohamed El Mankibi, and Amine Zoubir. "Review of natural ventilation models." *Energy Procedia* 78 (2015): 2700-2705. https://doi.org/10.1016/j.egypro.2015.11.355.
[6] Rundle, C. A., M. F. Lightstone, P. Oosthuizen, P. Karava, and E. Mouriki. "Validation of computational fluid dynamics simulations for atria geometries." *Building and Environment* 46, no. 7 (2011): 1343-1353. https://doi.org/10.1016/J.BUILDENV.2010.12.019.
[7] Zhiqiang Zhai, *Computational Fluid Dynamics Applications in Green Building Design*, 2014.
[8] Tan, Gang, and Leon R. Glicksman. "Application of integrating multi-zone model with CFD simulation to natural ventilation prediction." *Energy and Buildings* 37, no. 10 (2005): 1049-1057. https://doi.org/10.1016/J.ENBUILD.2004.12.009.
[9] Liping, Wang, and Wong Nyuk Hien. "The impacts of ventilation strategies and facade on indoor thermal environment for naturally ventilated residential buildings in Singapore." *Building and Environment* 42, no. 12 (2007): 4006-4015. https://doi.org/10.1016/J.BUILDENV.2006.06.027.
[10] Liang, Nan, Qiongxiang Kong, Ying Cao, Weirong Zheng, Qian Yang, and Lianying Zhang. "Thermal Performance of an Atrium with Hybrid Ventilation and cooled air supply." In *IOP Conference Series: Earth and Environmental Science*, vol. 268, no. 1, p. 012088. IOP Publishing, 2019. https://doi.org/10.1088/1755-1315/268/1/012088.
[11] Kamar, Haslinda Mohamed, Nazri Kamsah, and J. L. Kam. "Indoor air of a double-storey residential house in Malaysia." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 31, no.1 (2017): 11-18.
[12] Wang, Fan, and Abd Halid Abdullah. "Investigating thermal conditions in a tropic atrium employing CFD and DTM techniques." *International Journal of Low-Carbon Technologies* 6, no. 3 (2011): 171-186. https://doi.org/10.1093/ijlct/ctr005.
[13] Khan, Naghman, Yuehong Su, and Saffa B. Riffat. "A review on wind driven ventilation techniques." *Energy and buildings* 40, no. 8 (2008): 1586-1604. https://doi.org/10.1016/J.ENBUILD.2008.02.015.
[14] Holford, Joanne M., and Gary R. Hunt. "Fundamental atrium design for natural ventilation." *Building and environment* 38, no. 3 (2003): 409-426. https://doi.org/10.1016/S0360-1323(02)00019-7.
[15] Daghigh, Roonak, and Kamaruzzaman Sopian. "Effective ventilation parameters and thermal comfort study of air-conditioned offices." *American Journal of Applied Sciences* 6, no. 5 (2009): 943.