CHEMICAL ENGINEERING TRANSACTIONS

VOL. 83, 2021

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.cetjournal.it

Guest Editors: Jeng Shiun Lim, Nor Alafiza Yunus, Jiří Jaromír Klemeš

ISBN 978-88-95608-81-5; **ISSN** 2283-9216

DOI: 10.3303/CET2183032

Study of Fresh Air Supply Vent on Indoor Airflow and Energy Consumption in an Enclosed Space

Xinyou Ho[a], Wai Shin Ho[a,*], Keng Yinn Wong[b], Mimi Haryani Hassim[b], Haslenda Hashim[a], Zarina Ab Muis[a], Nor Alafiza Yunus[a], Gabriel Hoh Teck Ling[c]

[a]Process Systems Engineering Centre (PROSPECT), School of Chemical and Energy Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[b]Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[c]Faculty of Buid Environment and Surveying, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
hwshin@utm.my

Majority of residents in many urban and suburban areas, especially in Southeast Asia, have seen a significant increase in time spent indoors over the past decade. It is not unusual for residents in these areas spending 70 - 90 % of their lives indoor either in home or office, and often in poorly ventilated rooms. These rooms are usually outfitted with ductless air conditioning (also known as room air conditioner) with no supply of fresh air for ventilation. The build-up of indoor air pollutants such as CO_2, pathogens, and various volatile organic compounds in poorly ventilated room have been shown to be detrimental to both health and well-being its human occupants. This work aims to study the effect of introducing fresh air via a supply vent on the airflow within the room, while keeping track of the corresponding increase of energy consumption in the form of cooling capacity caused by ventilating warm fresh air into the room. To carry out this study, an air change per hour (ACH) of 10 supplied through different vent heights are simulated using a validated computational fluid dynamics model to investigate the behaviour of temperature, airflow, and energy consumption for five different vent-position cases. The simulation simulated case suggests that up to 27.7 % of a 1.0 horsepower (2.67 kW) air conditioner's cooling capacity may be required to maintain the room at 22 °C with noticeable difference in airflow pattern within the room depending on the vent position. These findings will serve as a guideline for practical applications in designing ventilation systems in a small-to-medium enclosed space.

1. Introduction

A study conducted by Neil et al. (2001) suggests that Americans spend, on average, 86.9 % of their time in enclosed buildings and only 7.6 % outdoors. In Southeast Asia, it is not unusual for residents in developed regions to spend 70 – 90 % of their time indoors either in office or home which are usually poorly ventilated. The proliferation of ductless air conditioning (or otherwise known as room air conditioner) among residential and small offices in recent years further exacerbate the problem by discouraging occupants from leaving doors and windows open for ventilation. Several studies have shown that indoor air pollutants such as CO_2 (Dorota and Bernadetta, 2018), volatile organic compounds, and fine particulate matter (Arindam et al., 2017) can quickly build up in such an environment. These build-up of indoor air pollutants are detrimental to human health and cognitive functions (Joseph et al., 2016). These studies stress the importance of ventilation for an enclosed space. ACH is often used to gauge the level of ventilation in a room. An ACH of 4 – 10 is usually recommended in most enclosed spaces apart from kitchens and clean rooms. However, due to the effects of stratification within the room, the position of the vent can drastically affect the air movement around the occupants (Yaming , 2017) and the overall effectiveness of the ventilation of the room. Ventilating fresh air directly from outside will also inevitably introduce additional heat energy in the form of sensible heat into the room which needs to be removed by the air conditioning system in order to maintain a constant temperature. Previous work such as Zhang and Li (2020) and Yang et al., (2019) successfully studied the airflow patterns and energy optimizations in rooms but did not include additional heat introduced into the system in their studies. This work aims to evaluate the

Paper Received: 15/07/2020; Revised: 22/08/2020; Accepted: 24/08/2020
Please cite this article as: Ho X., Ho W.S., Wong K.Y., Hassim M.H., Hashim H., Ab Muis Z., Yunus N.A., Ling G.H.T., 2021, Study of Fresh Air Supply Vent on Indoor Airflow and Energy Consumption in an Enclosed Space, Chemical Engineering Transactions, 83, 187-192 DOI:10.3303/CET2183032

effect of fresh air vent position on airflow, temperature, and energy consumption in order to minimize the energy required for ventilation in line with Ramli et. al (2020)'s work to reduce carbon emissions.

1.1 CFD simulations

Computational fluid dynamics (CFD) simulation is a powerful and effective method to predict the movement of air within the room caused by a fresh air vent. Most CFD simulations done to-date on indoor ventilation have been very successful in predicting the air flow of centralized air conditioning systems (Shan et al., 2019) where return air mixed with fresh air is cooled before ventilated back into a room. However, in this study, fresh air is ventilated directly into the room without cooling. The higher temperature of fresh air requires a new model based on existing CFD models to evaluate the indoor airflow and its impact on the energy consumption of the air conditioning system. Most indoor CFD simulations assume air to be incompressible, which may downplay the effect of density and buoyancy of hot air in convection currents within the room. The difference in air temperatures can also lead to stratification with varying effects on the indoor airflow (Krajčík et al., 2012). In order to investigate the velocity (*u*) and temperature profile of indoor airflow caused by different vent positions, a compressible flow of air with heat transfer model is investigated. As doors are identified the primary outlet of air in a room, two outlets are modelled as the top and bottom door gap. An estimate of the additional energy consumption due to the influx of sensible heat energy (h_s) can be computed by extracting the temperature and velocity data of the air leaving through both outlets which is included in this work.

2. Methodology

2.1 Development and validation of numerical simulation

OpenFOAM v7 is used to obtain a numerical solution for a buoyant, compressible, and turbulent flow of air. The modelling and meshing of geometry are done using FreeCAD with CfdOF addon; Reynold-Averaged Simulation (RAS) k-ε model is selected as the turbulence model; a variant of Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm for turbulent and buoyant flows called buoyantSimpleFoam is used as the solver; heRhoThermo with constant cp as thermophysical model; no-slip condition for wall boundary, and convergence criterion below 10^{-5}. Grid Sensitivity Analysis is also carried out to determine the discretisation error of the CFD model. A small discretisation error of less than 5 % indicates that the mesh grid is small enough and approaching the true numerical solution is performed by doubling the number of mesh grid until a discretisation error of less than 5 % is obtained. Verification of the numerical solution is done by recreating a validated CFD simulation study by Deng et al. (2018) for ACH of 10, 16, and 28. This recreated simulation is further split into two parts: isothermal and elevated inlet air temperature. The simulation for the elevated inlet air temperature (ΔT_{inlet} = 5 °C) is important to verify the effect of buoyancy and convection current in this numerical simulation. Table 1 below details the key differences between the simulations, and the validation of CFD_{new} simulation results with Deng et al. (2018)'s work in Figure 1a and Figure 2:

Table 1: Comparison of CFD models by Deng et al. (2018) and CFD_{new} model

Description	Deng et al. (2018)	New simulation, CFD_{new}
CFD Software	FLUENT ANSYS 16.0	OpenFOAM v7
Solver	SIMPLE	buoyantSimpleFoam
Flow type	Turbulent, incompressible	Turbulent, compressible
Equations	Continuity, momentum, contaminant and air transport	Continuity, momentum, air transport, heat
Turbulence model	RNG k-ε	RAS k-ε

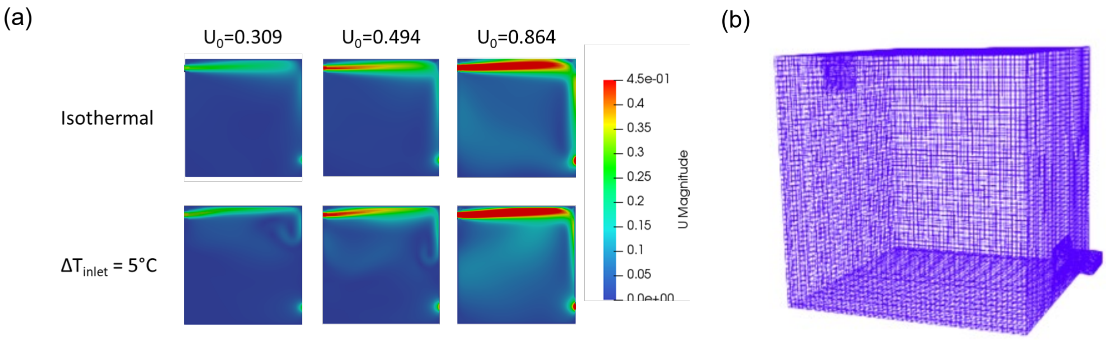

Figure 1: (a) CFD_{new} simulation validation results with Deng et al. (2018)'s work; and (b) CFD_{new} mesh.

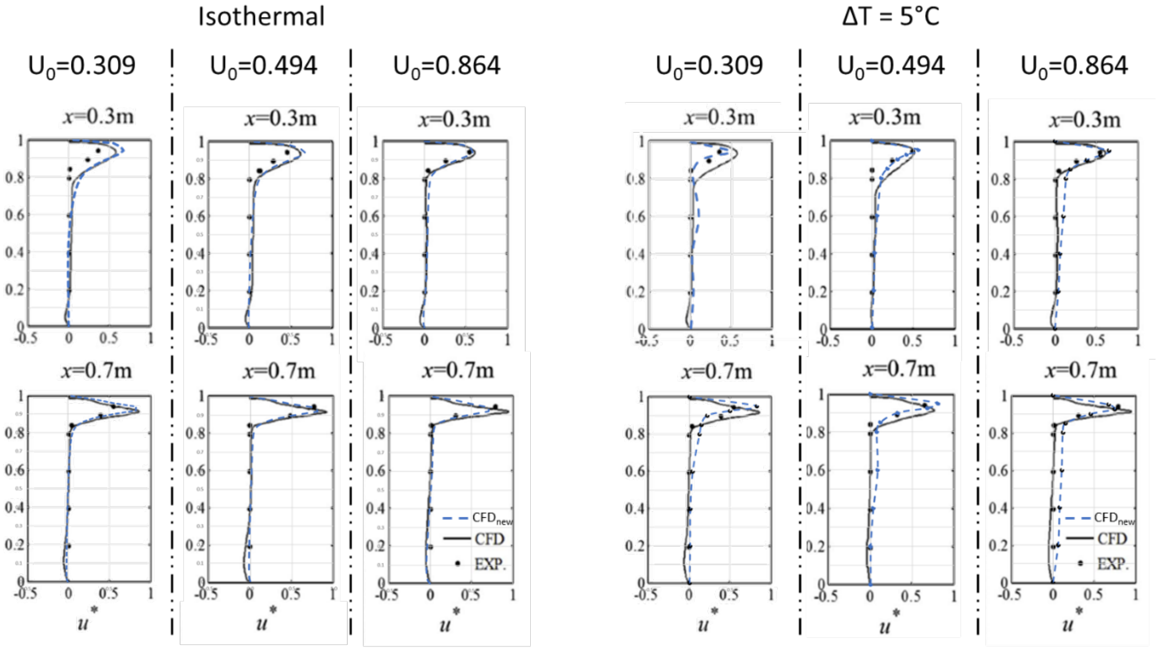

Figure 2: Comparison of CFDnew (isothermal and ΔT_{inlet} = 5 °C) simulations of normalized x-velocities ($u^* = u_x/u_0$) with Deng et al. (2018) simulated data and experimental results.

Data shown in Figure 1a and 2 shows CFDnew isothermal model is able to replicate the simulation by Deng et al. (2018) very closely. Analysing the simulation results for warm inlet air (ΔT_{inlet} = 5 °C) shows the simulation results move closer to the experimental results by Deng et al., (2018). This is likely caused by the lower temperature of CO_2 released in the middle of the enclosure as a pollutant source.

2.2 Geometry model and mesh of ventilated room

A room with dimensions of 3.65 m x 4.25 m x 2.75 m is modelled with an inlet opening of 0.2 m x 0.2 m. To simulate air leaving out of the room through a door opening, two outlets with dimensions of 0.05 m x 0.7 m are each placed at 0.05 m and 2.0 m high. Three CFD simulations are carried out for three vent heights of 0.05 m, 1.275 m, and 2.5 m. An ACH of 10, or 2.97 m/s, is selected as the inlet flow speed. The inlet air temperature is set at 305 K (32 °C) while the ambient air temperature in the room is 295 K (22 °C).

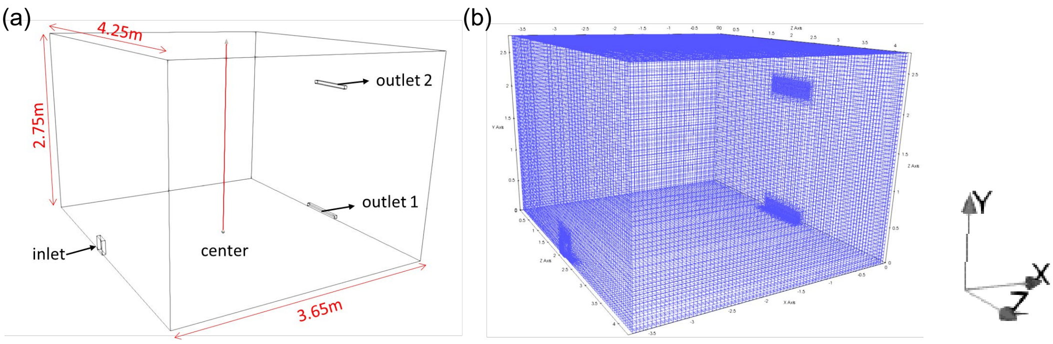

Figure 3: (a) Geometry and (b) mesh of simulated room with fresh air vent height = 0.05 m.

3. Results and discussion

3.1 Relationship between inlet vent height, indoor air flow, and temperature profile

By running the newly validated CFDnew model with mesh generated in Figure 3, velocity contours, velocity vectors, and temperature profiles of airflow within the simulated room for three vent positions are shown in Figure 4 below. Figure 5 charts the x-velocity profile along the y-axis at the geometrical center of the room as indicated in Figure 3a.

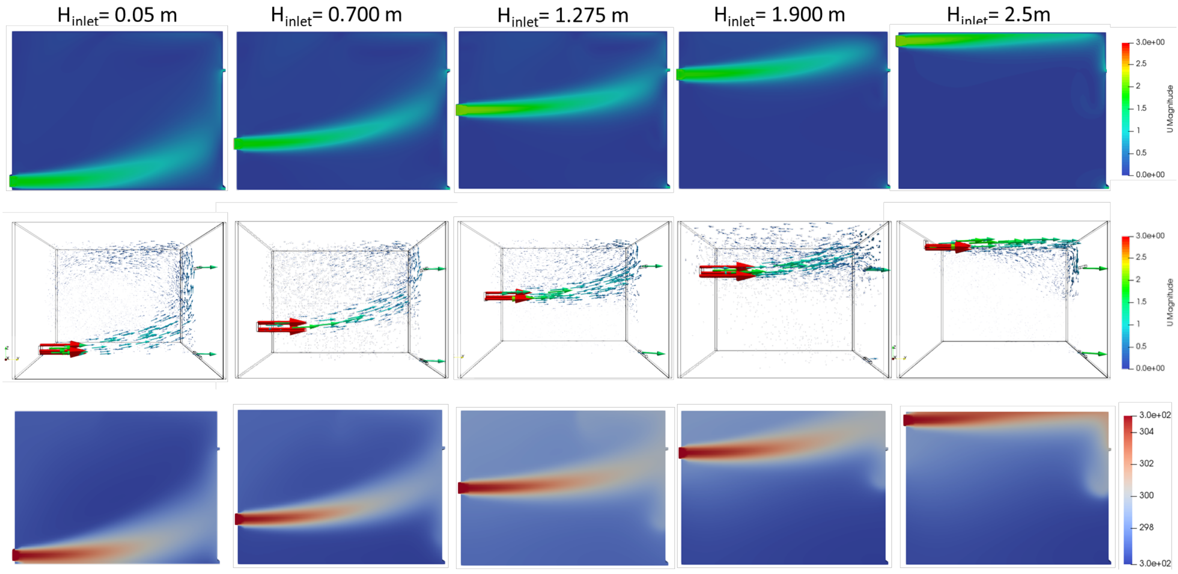

Figure 4: Comparison of velocity contour, velocity vector, and temperature profile of air at the xy-plane for different vent heights obtained using numerical simulations for ACH = 10.

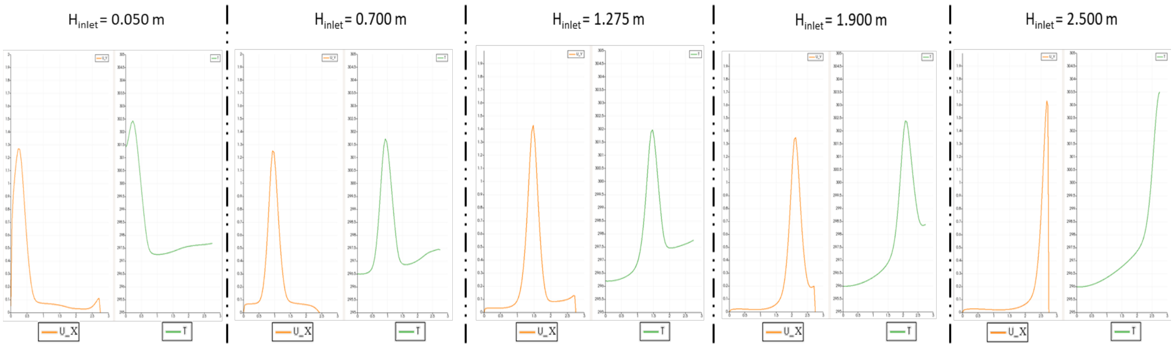

Figure 5: Air velocity along x-axis (u-x) and temperature (T) extracted along y-axis at the center of simulated room when ACH = 10.

According to the results obtained from the CFD simulation, changes in airflow at the center of the room corresponds mainly with the height of the inlet vent. Taking the average sitting height of an individual to be 1.3 m, and in the absence of any other form of mechanical ventilation, the airflow around any occupants sitting in the middle of the room is unsatisfactory for inlet vent height 2.5 m even with a generous ACH of 10. When the inlet vent height is lowered to 1.275 m, airflow around the occupant is greatly improved. The higher air temperature of 300.1 K (27.1 °C) moving across the occupant's breathing area may not provide adequate thermal comfort as recommended by American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE) 55. This vent height is also shown to heat up the room most evenly among the simulated vent heights. Placing the inlet vent near floor level at 0.05 m elevation generates a marginally better air flow (0.088 m/s) along the x-axis at 296.4 K (23.4 °C) for the occupant breathing area but leaves the feet area much warmer 301.4 K (28.1 °C). The high temperature difference between head and feet of 4.7 °C is more than 3 °C recommended value and may also cause discomfort to the occupant.

3.2 Heat transferred into system from fresh air ventilation

There are a few assumptions made when estimating the additional heat energy (or cooling load) required for the air conditioning to remove in order to maintain 295 K (or 22 °C) room temperature. These assumptions include: only sensible heat (h_s) is taken into consideration in this simulation, specific heat capacity of air, c_p is assumed constant at 1.006 kJ/kgK; density of ambient air, ρ is initially assumed 1.18 kg/m3; mass flow rate through the room is constant and conserved; and the simulated room is a closed system in thermal equilibrium.

The total heat energy transferred into the room can then be calculated by subtracting the heat leaving the room (T_{out}) from the heat entering the room (T_{in}). Table 2 lists the equations used to calculate the heat transfer:

Table 2: Summary of heat transfer equations used

Description	Equation	Reference
General heat transfer	$Q^{\cdot} = \dot{m} \cdot c_p \cdot \Delta T$	Holman (2010)
Sensible heat transfer	$h_s = \rho \cdot A \cdot u \cdot c_p (T_{in} - T_{out})$	Shan (2000)
Normalised h_s^*	$h_s^* = h_s \, / \, h_{s,inlet}$	

where $Q^{\cdot}$ represents heat transfer rate; $m^{\cdot}$ for mass flow rate; c_p for specific heat capacity of air; ΔT represents the difference in temperature between fresh air and indoor air; $h_{s,inlet}$ for sensible heat from inlet. Mass flow rate is calculated as a product of density (ρ), area of inlet/outlet (A), and velocity (u). In Figure 6a, normalised sensible heat, h_s^* is used to illustrate the difference in rate of heat transferred into the room on a common scale for fresh air vents of different heights and mass flow rates; while Figure 6b charts the difference in both normalized mass flowrate and average temperature for upper and lower outlets.

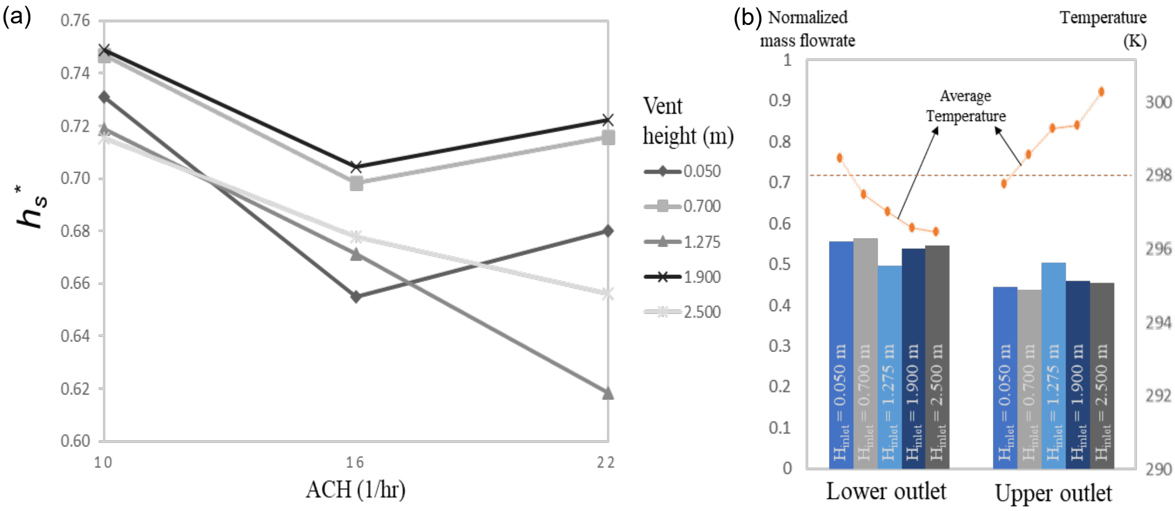

Figure 6: Comparison of (a) normalized sensible heat rate, h_s^* with different ACHs and vent heights; (b) normalized mass flowrate and temperature of upper and lower outlet for ACH = 10

By analyzing the simulation results, it appears to show that as ACH increases, vent height will be a bigger factor to the thermal efficiency of the ventilation system, albeit minimal. It is thought that the larger ACH may cause either increased stratification – leading to a higher T_{out} of air leaving via upper outlet; or increased mixing between hot and cold air – causing more heat to remain in the room depending on the vent height relative to the outlets. From Figure 6b, air is shown to exit the lower outlet in higher proportion compared to the upper outlet. This is expected to be caused by the higher density of cooler air and the tendency for hot air to rise upwards. The temperature of air leaving at the lower outlet is also noticeably lower regardless of vent height. As there seems to be some stratification effect, especially when the vent height is placed 2.5 m above floor level, it is possible to reduce the cooling capacity required by the air conditioning system to maintain a constant temperature by providing an outlet near ceiling level. Taking just the ventilation rate in ACH and heat transfer into consideration, a fresh air vent placed close to the ceiling (2.5 m in this case) yields good cost-to-ventilation performance ratio reliably due to stratification. This configuration, however, falls short in providing adequate air movement around the occupant's breathing area. By lowering the vent height, air flow around the occupant improves and but more heat is transferred into the room and the temperature around the occupant increases accordingly. Surprisingly, a low vent height of 0.05 m is found through simulation to serve as a decent alternative to achieve a small air movement (0.088 m/s), good thermal comfort (23.4 °C), and decent thermal efficiency (h_s^* = 0.73) at 10 ACH. It should also be noted the outlets are placed directly opposite of the fresh air vent in this simulation and this might lower the mixing of warm and cold air which results in higher temperature of outlet air. Assuming a 1.0 horsepower (or 2.67 kW cooling capacity) ductless air conditioner is used to cool the simulated room, a vent with a height of 0.7 m and 10 ACH is simulated to add 0.739 kW (or 27.7 % cooling capacity of 1.0 hp air conditioner) of heat into the room while a 2.5 m high vent will add 0.706 kW (26.4 % cooling capacity).

4. Conclusion

In this work, the study focuses mainly on the inlet ventilation rate and its effect on airflow, temperature profile, and energy consumption in the form of cooling capacity required to maintain a constant temperature in a room. As people spend more time in poorly ventilated rooms, more attention is required in this area to look for more effective and efficient ventilation. From the simulation carried out in this study, inlet vent height appears to have some effect on the rate of heat transfer into a room (Δh_s^* up to 4.2 %, 7.0 %, and 14.3 % for 10, 16, 22 ACH) for this ventilation configuration. There is also evidence for thermal stratification, especially when the inlet vent height is close to ceiling level, presenting consistently across all 10, 16, and 22 ACHs. The presence of thermal stratification within a room may be beneficial if a ventilation and air conditioning system can be designed to take advantage of it. For example, this simulation indicates that with ventilation at 2.5 m and 10 ACH, a ductless air conditioning system is best installed at a height of 2 m where the air temperature is 24.3 °C instead of 2.6 m (air temperature 28.1 °C). By doing so the air conditioning system will be able to lower its energy consumption by cooling a smaller volume of colder air within the room while disregarding most of the warmer air near ceiling level and reduce the cost required to provide ventilation. While not included in this work, other studies suggest that heavier indoor pollutants such as CO_2 tends to collect at the bottom of the room and can be removed effectively when ventilated out at a low outlet height. Combined with the findings in this work, fresh air ventilation at floor level may be a decent alternative especially if the temperature difference between the fresh air and indoor air is small. Future works to include occupant modelling and optimization strategies for venting fresh air may uncover great benefits to reduce overall energy required for ventilation. These findings will serve as a good reference when considering design of efficient and effective ventilation system for small-to-medium rooms.

Acknowledgments

The authors would like to acknowledge Universiti Teknologi Malaysia for the research grants with vot number of 02M03, 00L51, 4C298, and 06G47 that was provided for this research study.

References

Arindam D., Suresh R., Akansha G., Damini S., Priyanka K., 2017, Indoor air quality of non-residential urban buildings in Delhi, India, Gulf Organization for Research and Development, 6, 412-120.

Deng, H.-Y., Feng, Z., Cao, S.-J., 2018, Influence of air change rates on indoor carbon dioxide stratification in terms of Richardson number and vorticity, Building and Environment, 129, 74-84.

Dorota A.K., Bernadetta W., 2018, Analysis of Indoor Air Parameters in an Educational Building, Energy Procedia, 147, 96–103.

Joseph G.A., Piers M., Usha S., Suresh S., Jose V., John D.S., 2016, Associations of Cognitive Function Scores with Carbon Dioxide, Ventilation, and Volatile Organic Compound Exposures in Office Workers: A Controlled Exposure Study of Green and Conventional Office Environments, Environmental Health Perspectives, 124, 805-812.

Holman J.P., 2010, Heat Transfer, 10th Edition. MacGraw-Hill, New York, USA.

Krajčík, M., Simone, A. and Olesen, B. W., 2012, Air distribution and ventilation effectiveness in an occupied room heated by warm air, Energy and Buildings, 55, 94-101.

Neil E. K., William C.N., Wayne R.O., John P.R., Andy M.T., Paul S., Joseph V.B., Stephen C.H., William H.E., 2001, The National Human Activity Pattern Survey (NHAPS): a resource for assessing exposure to environmental pollutants', Journal of Exposure Science & Environmental Epidemiology, 11, 231–252.

OpenFOAM V7, 2020, The OpenFOAM Foundation Ltd, Berkshire, United Kingdom.

Ramli A. F., Idris A. M., Ab Muis Z., Ho W. S., 2020, Multi-stage Carbon Emission Pinch Analysis of an Integrated Power Generation and Electric Vehicle System, Chemical Engineering Transactions, 78, 457-462.

Schibuola L, Scarpa M, Tambani C., 2016, Natural ventilation level assessment in a school building by carbon dioxide concentration measures, Energy Procedia, 101, 257–64.

Shan K. W., 2000, Handbook of Air Conditioning and Refrigeration, 2nd Edition. MacGraw-Hill, New York, USA.

Shan X., Xu W., Lee Y.-K. Lu W.-Z., 2019, Evaluation of thermal environment by coupling CFD analysis and wireless-sensor measurements of a full-scale room with cooling system, Sustainable Cities and Society, 45, 395-405.

Yang Li, Liu X., Qian F., Du S., 2019, Ventilation effect on different position of classrooms in “line” type teaching building, Journal of Cleaner Production, 209, 886-902.

Yaming F., Xiandong L., Yihuan Y., Jiyuan T., 2017, Overall performance evaluation of underfloor air distribution system with different heights of return vents, Energy and Buildings, 147, 176-187

Zhang X., Li H., 2020, Feasibility analysis for control of bioaerosol concentration at indoor corner via airflow from ventilation outlet with energy optimization, Journal of Cleaner Production, 248, 1-14.