Study of Effects of Inlet Wind Velocity and Direction on Airflow around the Buildings Using CFD Turbulence Models: A Case Study of Rajamangala University of Technology Rattanakosin (Salaya Campus), Thailand

Jirapol Klinbun[1], Tipapon Khamdaeng[2], and Numpon Panyoyai[3]

[1]Department of Mechanical Engineering, Faculty of Engineering, Rajamangala University of Technology Rattanakosin Salaya Campus, Nakhon Pathom, Thailand
[2,3]Faculty of Engineering and Agro-Industry, Maejo University, San Sai, Chiang Mai, Thailand
E-mail: jirapol9@hotmail.com, tipapon@mju.ac.th, numpon@mju.ac.th

Received: Feb 8, 2022 / Revised: March 21, 2022 / Accepted: April 18, 2022

Abstract—**This research investigates of the effects of inlet wind velocity and inlet wind direction on airflow around the buildings of Rajamangala University of Technology Rattanakosin (Salaya), Thailand using Computational Fluid Dynamics (CFD) turbulence models. The evaluation of a CFD model's performance and validation of its predictions with high-quality experimental data is necessary before the model is used in practice. In this study, there are 2 Models. Model 1 is the simulation for an inlet wind velocity of 0.5 and 1.5 m/s in the west direction. Model 2 is the simulation for an inlet wind velocity of 0.5 and 1.5 m/s in the south direction. The results were found that the higher inlet wind speeds at the inlet flow boundary would lead to a higher increase in the average speed of air around the building. In addition, the combined wind speed and inlet flow direction were affected to where the maximum wind speed occurs. The data obtained from this study will serve as a future basis for the construction of buildings in the university to provide better natural ventilation.**

Index Terms—**Airflow, Numerical Model, Wind Speed, Wind Direction**

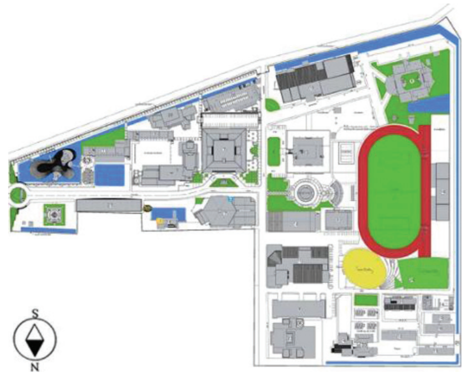
I. Introduction

The university land area which is consisting of a lecture building, operating buildings, offices, stadiums, libraries, etc. They should have open spaces for outdoor activities, and the area should also be good air ventilation. Surrounding infrastructure and buildings play an important role in the airflow around the buildings. In addition, controlling the direction of air and the speed of air flowing into the building and the living space surrounding the building provides a consistent wind speed and is suitable for activities in that part of the area [1]- [3]. Current analysis of airflow dynamics surrounding the building has become an integral part of the planning of the project or construction. This is because the energy of the airflow is an important factor that contributes to thermal comfort both indoors and outdoors [4]-[7]. The study of airflow around the building simulation is a new matter to help in this area. It can be performed in two ways: (a) physical model and (b) computer simulation. By the way, the computer simulation will yield faster results and a lower budget than the actual building model. However, the right choice and compromise between the accuracy and cost associated with modeling wind flow around buildings is essential in meeting practical engineering needs. The airflow around buildings has been studied for several purposes, such as (a) determination of wind surface pressure distribution on building envelopes, (b) turbulent dispersion of airborne contaminants, and (c) pedestrian comfort. There are three widely used numerical approaches for analyzing outdoor turbulent flow using Computational Fluid Dynamics (CFD) based on Reynolds Averaged Navier-Stokes (RANS) equation, Large Eddy Simulation (LES), and Direct Numerical Simulation (DNS) [8]-[10]. Many previous researchers have studied the flow around a single building (bluff body) or multiple buildings [7], [11]-[15]. The literature review revealed that there are limited studies available for intermediate cases, in which the buildings are close enough and the flow around the building is strongly affected by adjacent buildings.

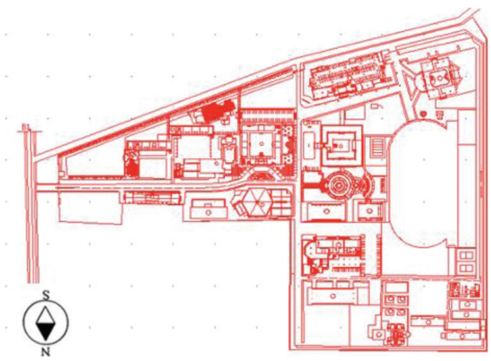
The main objective of the present study is to provide a faster, yet reliable and simple modeling tool for simulations of outdoor airflow around multiple buildings. The intent is to improve the 2D k-ε turbulence models mainly developed and used for modeling airflow in indoor environments. The measurement airflow around the building at the Rajamangala University of Technology Rattanakosin (Salaya campus) has been carried out to provide the base for input data in the simulation. Then, the effects of inlet wind velocity and direction on airflow around the buildings were obtained.

II. Material and Methods

The Rajamangala University of Technology Rattanakosin (Salaya campus) has been selected for investigation of the airflow around the buildings. There are 25 buildings in the area, as seen in Fig.1. In order to fit the model with the similarity analysis of incoming flow, modeled buildings have been scaled to a 1:100 ratio.



(a)



(b)

Fig. 1. Modeled buildings in area of the Rajamangala University of Technology Rattanakosin (Salaya campus) (Ref. https://building.rmutr.ac.th)

A. Numerical Models

For a systematic study of the effects of wind inlet direction and wind speed on airflow, the application of computational fluid dynamics offers considerable advantages over field measurements and wind-tunnel experiments [16]. However, a main disadvantage of CFD is that it can be computationally expensive when increasing the resolution of the computational mesh and/or the size of the computational domain [17]. From Fig. 1, a model is drawn to calculate the airflow around the building as seen in Fig. 2.

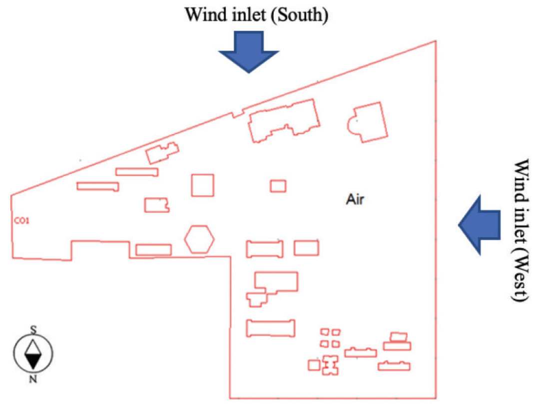


Fig. 2. Calculation domain

In this study, a numerical model is developed using COMSOL Multiphysics (Version 4.4), which is CFD software. The details of the numerical model including governing equations, turbulence modeling, boundary conditions, and initial condition are presented on the following topics.

Assumption: (a) Flow field is two-dimensional and steady. (b) All thermal properties are constant. (c) Air is an incompressible fluid.

Continuous equation [18]:

$$\nabla \cdot U = 0 \tag{1}$$

Navier–Stokes equations [18]:

$$\rho U \cdot \nabla U = \nabla \cdot \left[\left(\eta + \rho \frac{c_\mu k^2}{\sigma_k \varepsilon} \right) \cdot \left((\nabla U) + (\nabla U)^T \right) \right] - \nabla P \tag{2}$$

Where ρ is the fluid density [kg/m^3], U is the average velocity [m/s], η is dynamics with a viscosity [Ns/m^2] P is pressure [Pa], k is the kinetic energy of turbulence [m^2/s^2], ε is the diffusion rate of the kinetic energy of the turbulence [m^2/s^3].

The k-ε turbulence model:

$$\rho U \cdot \nabla k = \nabla \cdot \left[\left(\eta + \rho C_\mu \frac{k^2}{\varepsilon} \right) \nabla k \right] + \frac{1}{2} \rho C_\mu \frac{k^2}{\varepsilon} \left((\nabla U) + (\nabla U)^T \right)^2 - \rho\varepsilon \tag{3}$$

The equation can be distributed as follows:

$$\rho U \cdot \nabla \varepsilon = \nabla \cdot \left[\left(\eta + \rho C_\mu \frac{k^2}{\varepsilon} \right) \nabla \varepsilon \right] + \frac{1}{2} \rho C_{\varepsilon 1} k \left((\nabla U) + (\nabla U)^T \right)^2 - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{4}$$

Where σ_k = 1.00, σ_ε = 1.30, $C_{\varepsilon 1}$ = 1.44, $C_{\varepsilon 2}$ = 1.92, C_μ = 0.09 are the standard values proposed by Spalding and Launder [19].

B. Boundary and Initial Conditions

All calculation setting is shown in Table I.

TABLE I
CALCULATION CONDITIONS

Inlet:	
$u=u_{input}$ m/s	$v=0.0$ m/s
$u=u_0$	
$k=\left(3I_T^{\,2}/2\right)\left(u_0\cdot u_0\right), \varepsilon=C_\mu^{\,0.75}\left(\left(3I_T^{\,2}/2\right)\left(u_0\cdot u_0\right)\right)^{1.5}/L_T$	
Outlet:	
$P=0$ Pa	
$\rho=\rho_0$	
$n\cdot\nabla k=0, n\cdot\nabla\varepsilon=0$	
Side wall:	
$n\cdot u=0,$	
$\left[\left(\eta+\eta_T\right)\left(\nabla u+\left(\nabla u\right)^T\right)\right]n=\left[\rho C_\mu^{\,0.25}k^{0.5}/\left(\ln\left(\delta_w^{\,+}\right)/K+C^+\right)\right]u$	
$n\cdot\nabla k=0,$	
$\varepsilon=C_\mu^{\,0.75}k^{1.5}/\left(K\delta_w\right)$, where $\delta_w^{\,+}=\delta_w\rho C_\mu^{\,0.25}k^{0.5}/\eta$	

For the inlet velocity (u_{input}), rotary vane anemometry is well-known as a measurement technique with good spatial and time resolution and has been established as a reliable and versatile technique for velocity measurements in flow fields with low to moderate turbulence intensities. Fig. 3 is shown the Testo 435 rotary vane anemometer which is an accuracy of ±0.3 °C. From the measurements, all the data are collected as preliminary data to simulate airflow.

Fig. 3. Rotary vane anemometer (Testo Model 435)

C. Numerical Methodology

The CFD simulation setup provides a summary of the computational domain description used in the analysis along with the grid distribution, boundary condition settings, and numerical scheme [20], [21]. A grid sensitivity test was conducted to ensure the computational accuracy of the simulations with COMSOL™ Multiphysics. As shown in Fig. 4, a structured grid is deployed to discretize the computational domain for COMSOL simulation. Two-dimensional structured mesh along with the finite element method were utilized to discretize the computational domain and to describe the mass and momentum transport for each cell. There are 167,728 elements in the calculation domain.

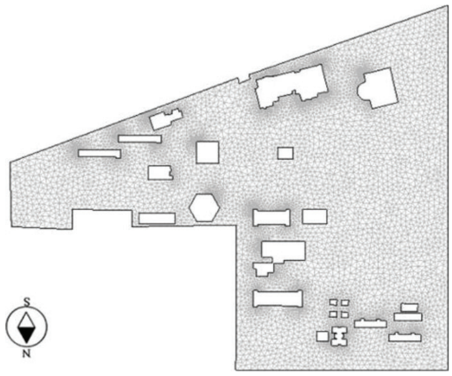


Fig. 4. Mesh generation

Fig.5 Shows the process of CFD simulation.

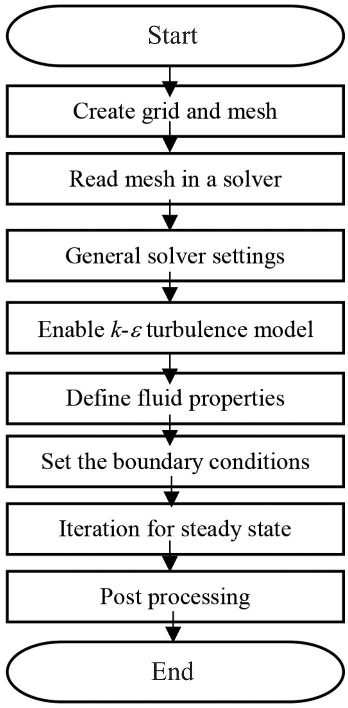


Fig. 5. Solution procedure for CFD model

III. Result and Discussion

The effects of inlet wind velocity and inlet wind direction on airflow patterns around the building in the Rajamangala University of Technology Rattanakosin (Salaya campus) are investigated using Computational Fluid Dynamics techniques. Fig.6 is shown the position and name of the building in the area.

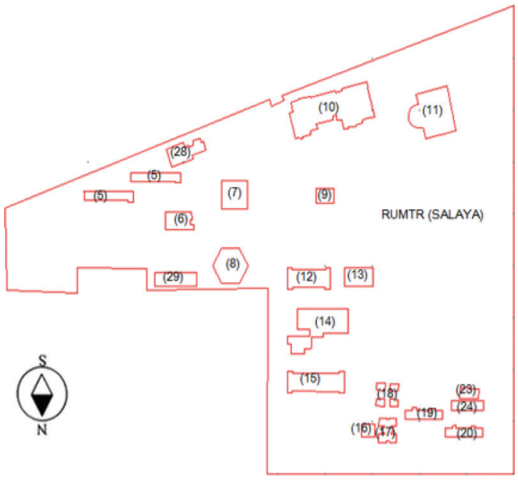
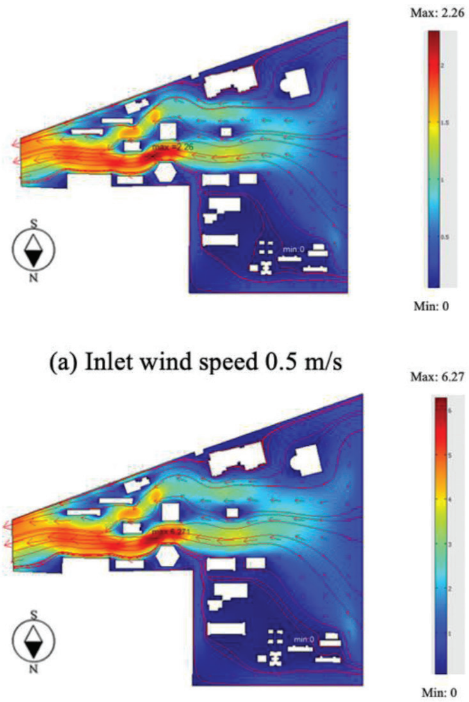


Fig. 6. The building location and name in the area of the Rajamangala University of Technology Rattanakosin (Salaya campus)

Where the buildings name according to Fig. 6 are as following:
(5) Building 1 and Building 2, Academic Promotion and Registration Office
(6) Faculty of Business Administration Building
(7) Office of the President
(8) Canteen
(9) Office of Service Science and Information Technology
(10) Sirindhorn Building
(11) Building for the 6th anniversary of the Royal Patronage
(12) Meeting room 1,500 seats
(13) Physical Education Building
(14) Faculty of Engineering Building
(15) Product design work building
(16) Office of training dormitory
(17) Training dormitory
(18) Directorate of Director General, 4 buildings
(19) Government Residential Building 1 (4 floors)
(20) Government Residential Building 2 (4 floors)
(23) House, building, location
(24) Operating Building, Premises

A. Model 1: Inlet Wind Velocity of 0.5 m/s and 1.5 m/s, Inlet Wind Direction in the West.

Fig.7 is shown the simulated air velocity pattern of model 1. The air inlet is from west (behind at university) to east (in front of the university) horizontal flow at speeds of 0.5 and 1.5 m/s respectively. Both cases (velocity 0.5 and 1.5 m/s) are based on the same airflow profiles around the buildings. The results show that when the airflow through and/or hits buildings, it affects the speed and pattern of the velocity. However, the rate of change increases with increasing inlet wind speed. From Fig. 7, the area in front of the canteen is the magnitude of velocity maximum. This is because the venturi effect causes the airflow to speed up. The high air speed is causing dust dispersion.



(a) Inlet wind speed 0.5 m/s

(b) Inlet wind speed 1.5 m/s

Fig. 7. The velocity profile around the building, the wind direction in the west

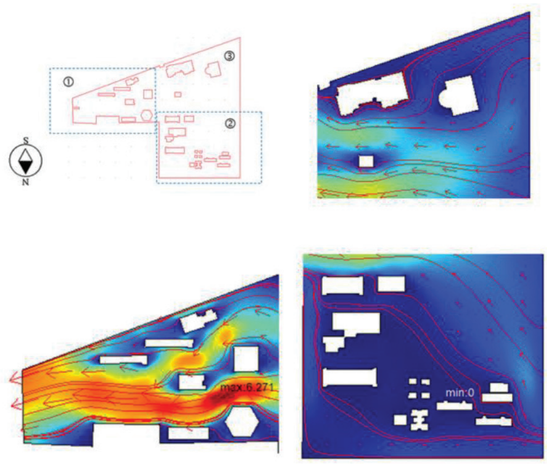


Fig. 8. The velocity profile around the building in each section, at an inlet wind speed of 1.5 m/s from west to east

Fig. 8 is shown the velocity profile in each section at a wind speed of 1.5 m/s from west to east. It is found that the comfort area (velocity about 1.0-2.0 m/s) occurs at the Office of Service Science and Information Technology. While airflow around the Government Residential Building 1 (No. 19) is the low velocity (occurring negative pressure) because there are many buildings in the area. The air velocity is less than 0.25 m/s.

B. Model 2: Inlet Wind Velocity of 1.5 m/s, Inlet Wind Direction in the South.

Based on the results of the CFD simulation, Fig. 9 is shown the velocity profile around the building, wind speed of 1.5 m/s, and the wind direction from south to north. It is found that the magnitude of velocity occurs between 1.5-2.0 m/s. The location of buildings No. 20, 23, and 24 with a wind speed of about 2.0-2.5 m/s, and air velocity is quite suitable to designate as a rest area. It is observed that the velocity of the air increases as it flows through the openings between the buildings.

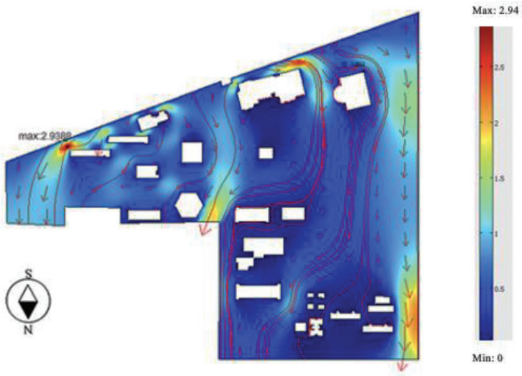


Fig. 9 The velocity profile around the building, the wind direction in the south

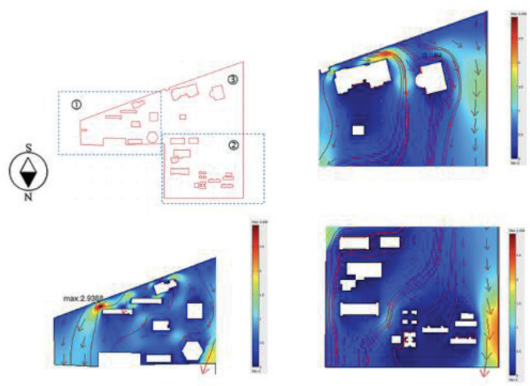


Fig.10. The velocity profile around the building in each section, at an inlet wind speed of 1.5 m/s from south to north

Fig. 10 is shown the velocity profile in each section at a wind speed of 1.5 m/s from south to north. It was found that the air velocity was about 2.9 m/s in the area behind Building 1 and Building 2 (No.5), which are the Academic Promotion and Registration Office, respectively. This is because of air hitting the buildings. While the negative pressure area occurs at the back of the 6th cycle of the King's Birthday (No. 11).

IV. Conclusions

This research was to study a numerical study of airflow around the building area of Rajamangala University of Technology Rattanakosin (Salaya) to increase the knowledge and understanding of the behavior of airflow around the building where the airflow enters in different directions.

The research begins with studying the basic knowledge of airflow, such as turbulent flow by using computational fluid dynamics. Then, a combination of the knowledge of numerical methodology including knowledge of airflow through various shapes and information on the fluid properties are used in the calculation. Finally, the nature of the airflow, when the wind blows in different directions and changes the speed is studied. The results of the research are shown for two models. For model 1, the comfort zone occurs around the Bureau of Science, Service, and Information Technology building (No. 9) with wind speeds of 1-2 m/s from west to east. For model 2, the comfort zone occurs around the Residential Building (No.20, 23, 24) These results are to obtain basic information about airflow and is the basis for the development of future mathematical models. This research is used as a tool for decision-making in building layout design within the area of Rajamangala University of Technology Rattanakosin (Salaya campus) in terms of energy saving.

V. Suggestion and Recommendation

For the next research, the models should analyze heat transfer and consider it three-dimensional to get results that are closer to the real phenomena.

The comparison magnitude of velocity between the simulation and experimental results should be carried out.

Acknowledgment

This research was conducted with the support of Rajamangala University of Technology Rattanakosin and research funding from the Thai national budget (A11/2560).

References

[1] E. Trepci, P. Maghelal, and E. Azar, "Urban Built Context as a Passive Cooling Strategy for Buildings in a Hot Climate," *Energy and Buildings*, vol. 231, p. 110606, Jan. 2021.

[2] N. Ba Chien, N. Viet Dung, T. Quoc Dung et al., "CFD Simulation of Multi-Outdoor Unit Configuration Design for a Building," *IOP Conference Series: Earth and Environmental Science*, vol. 505, no. 1, p. 12007, Jul. 2020.

[3] A. Izadi, M. Rudd, and V. M. Patrick, "The Way the Wind Blows: Direction of Airflow Energizes Consumers and Fuels Creative Engagement," *Journal of Retailing*, vol. 95, pp. 143-157, Sep. 2019.

[4] Q. Yi, M. Konig, D. Jank et al., "Wind Tunnel Investigations of Sidewall Opening Effects on Indoor Airflows of a Cross-Ventilated Dairy Building," *Energy and Buildings*, vol. 175, pp. 163-172, Jul. 2018.

[5] X. Zhang, A. U. Weerasuriya, and K. T. Tse, "CFD Simulation of Natural Ventilation of a Generic Building in Various Incident Wind Directions: Comparison of Turbulence Modeling, Evaluation Methods, and Ventilation Mechanisms," *Energy and Buildings*, vol. 229, p. 110516, Dec. 2020.

[6] G. K. Ntinas, X. Shen, Y. Wang et al., "Evaluation of CFD Turbulence Models for Simulating External Airflow Around Varied Building Roof with Wind Tunnel Experiment," *Building Simulation*, vol. 11, no. 1, pp. 115-123, Apr. 2018.

[7] W. Zuo and Q. Chen, "Real-Time or Faster-Than-Real-Time Simulation of Airflow in Buildings," *Indoor Air*, vol. 19, no. 1, pp. 33-44, Feb. 2009.

[8] F. Ascione, R. F. de Masi, M. Mastellone et al., "The Design of Safe Classrooms of Educational Buildings for Facing Contagions and Transmission of Diseases: A Novel Approach Combining Audits, Calibrated Energy Models, Building Performance (BPS) and Computational Fluid Dynamic (CFD) Simulations," *Energy and Buildings*, vol. 230, p. 110533, Jan. 2021.

[9] J. K. Calautit, D. O'Connor, P. W. Tien et al., "Development of a Natural Ventilation Windcatcher with Passive Heat Recovery Wheel for Mild-Cold Climates: CFD and Experimental Analysis," *Renewable Energy*, vol. 160, pp. 465-482, Nov. 2020.

[10] M. M. Hefny and R. Ooka, "Influence of Cell Geometry and Mesh Resolution on Large Eddy Simulation Predictions of Flow Around a Single Building," *Building Simulation*, vol. 1, no. 3, pp. 251-260, Sep. 2008.

[11] R. Ramponi and B. Blocken, "CFD Simulation of Cross-Ventilation for a Generic Isolated Building: Impact of Computational Parameters," *Building and Environment*, vol. 53, pp. 34-48, Jul. 2012.

[12] K. H. Cheong, Y. H. Teo, J. M. Koh et al., "A Simulation-Aided Approach in Improving Thermal-Visual Comfort and Power Efficiency in Buildings," *Journal of Building Engineering*, vol. 27, p. 100936, Jan. 2020.

[13] X. Tong, S. W. Hong, and L. Zhao, "CFD Modeling of Airflow, Thermal Environment, and Ammonia Concentration Distribution in a Commercial Manure-Belt Layer House with Mixed Ventilation Systems," *Computers and Electronics in Agriculture*, vol. 162, pp. 281-299, Jul. 2019.

[14] M. F. Yassin, "Numerical Study of Flow and Gas Diffusion in the Near-Wake Behind an Isolated Building," *Advances in Atmospheric Sciences*, vol. 26, no. 6, pp. 1241-1252, Nov. 2009.

[15] M. Ning, S. Mengjie, C. Mingyin et al., "Computational Fluid Dynamics (CFD) Modelling of Air Flow Field, Mean Age of Air and CO_2 Distributions Inside a Bedroom with Different Heights of Conditioned Air Supply Outlet," *Applied Energy*, vol. 164, pp. 906-915, Feb. 2016.

[16] M. S. Al-Homoud, "Computer-Aided Building Energy Analysis Techniques," *Building and Environment*, vol. 36, no. 4, pp. 421-433, May. 2001.

[17] Q. Cheng, H. Li, L. Rong et al., "Using CFD to Assess the Influence of Ceiling Deflector Design on Airflow Distribution In the Hen House with Tunnel Ventilation," *Computers and Electronics in Agriculture*, vol. 151, pp. 165-174, Aug. 2018.

[18] C. Porras-Amores, F. R. Mazarrón, I. Cañas et al., "Natural Ventilation Analysis in an Underground Construction: CFD Simulation and Experimental Validation," *Tunnelling and Underground Space Technology*, vol. 90, pp. 162-173, Aug. 2019.

[19] B. E. Launder and D. B. Spalding, "The Numerical Computation of Turbulent Flows," *Computer Methods in Applied Mechanics and Engineering*, vol. 3, no. 2, pp. 269-289, Mar. 1974.

[20] S. J. G. Shoba and A. B. Therese, "Detection of Glaucoma Disease in Fundus Images Based on Morphological Operation and Finite Element Method," *Biomedical Signal Processing and Control*, vol. 62, p. 101986, Sep. 2020.

[21] N. Mao, M. Song, D. Pan et al., "Computational Fluid Dynamics Analysis of Convective Heat Transfer Coefficients for a Sleeping Human Body," *Applied Thermal Engineering*, vol. 117, pp. 385-396, May. 2017.

Jirapol Klinbun was born in Lopburi Province, in 1973. He received the B.S. degree in mechanical engineering from Rajamangala University of Technology Thanyaburi, Pathumthani, Thailand in 1994. The M.S. and Ph.D. degree in mechanical engineering from Chiang Mai University, Thailand, in 2000 and 2008. From 2006 to 2008, he was a Research Assistant with the Heat Pipe Laboratory.

Since 2021, he has been an Associate Professor with the Mechanical Engineering Department, at Rajamangala University of Technology Rattanakosin. His research interests include Thermo-Fluid, Heat transfer in a heat pipe, Energy saving, Automotive technology, and Hydraulics-Pneumatics system.

Tipapon Khamdaeng received a B.Eng. degree in food engineering from Maejo University and a Ph.D. degree in mechanical engineering from Chiang Mai University, in 2006 and 2012. Since 2016, she has been an Assistant Professor with the Mechanical Engineering, Faculty of Engineering and Agro-Industry, Maejo University. Her main research interests include the computational modeling and simulation of solid and fluid mechanics and heat transfer characteristics of biomaterials

Numpon Panyoyai received a B.E. degree in mechanical engineering from Rajamangala University of Technology Lanna, Chiang Mai, Thailand, in 2000. He received an M.E. and D.E. degree in mechanical engineering from Chiang Mai University, Thailand, in 2005 and 2013.

Since 2012, he has been an Assistant Professor with the Mechanical Engineering, Faculty of Engineering and Agro-Industry, Maejo University. His research interest includes Mechanical and machinery design, biomass and biochar technology, thermal systems, and heat exchangers.