

ANALYSIS OF PARTICLE FLOWS (PM1, PM5, PM10) IN FORCED VENTILATION SYSTEM USING CFD TECHNIQUE

Machimontorn Promtong^{*}, Pannita Phirommark, Ratchanon Siri, Phusit Kaewsawatwong, Jetsadaporn Priyadumkol and Chakrit Suvanjumrat

Received: August 30, 2021; Revised: March 07, 2022; Accepted: March 26, 2022

Abstract

About 80% of the time, people living indoors face an unacceptable quantity of tiny particles that can affect respiration systems and lead to serious health issues. This research objective focuses on investigating forced ventilation systems and how the crucial parameters relating to the particle, including the size, inlet velocity, and room ventilating system, can influence the movement of harmful particles. The Computational Fluid Dynamics (CFD) technique was introduced to investigate a room model's air movements and compared with experimental data. Due to the turbulent flow, two different turbulence models ($k-\varepsilon$ and $k-\omega$ SST) were involved and have been discussed when calculating the flow field, as in the Reynolds stress closures. The simulated velocity profiles were found in good agreement with experimental data. To obtain additional information on particle propagations, the Discrete Phase Model (DPM) was introduced. It was found that flow patterns can dominate particle motions in the considered ventilation systems dependent on the size and inlet velocity. Due to having different inlet velocities, the number of escaped particles and the residence times were found to represent different values for each particle size. As a result, the $1\ \mu\text{m}$ particles could completely escape from the room within 13 min, but the larger $10\ \mu\text{m}$ remained in the room for longer, finally escaping after 16 min. This DPM modeling technique will be further introduced to study such infected particles' movements in commercial or public places. Modifications to enhance the ventilation system's efficiency in particle removal will be crucial in discussions.

Keywords: Computational Fluid Dynamics (CFD), Discrete Phase Model (DPM) modelling, Particle tracking and movement, Forced ventilation system

Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, 73170, Thailand. E-mail: machimontorn.pro@mahidol.edu

^{*}Corresponding author

Introduction

Small clouds of dust or harmful particles may be created by pollution from vehicles and industrial activities which becomes suspended in the atmosphere. Likewise, the burning of crops in the fields can produce smoke, as can burning incense, so there is an increase of harmful particles, including PM1 and PM10, in the atmosphere (W.H.O., 2021). Occasionally, dust particles can travel into a building and become trapped inside the building if there is an imperfect ventilation system. Such particle matter can have a massive impact on respiratory health, causing illnesses such as lung disease, allergies, and Sick Building Syndrome (SBS) (Lab, 2021). Similarly, liquid droplets can become a virus carrier (COVID-19); therefore, spreading diseased particles which could lead to viral infection (Vuorinen *et al.*, 2020, W.H.O., 2020; Ren *et al.*, 2021). Ventilation system design is crucial to the removal of harmful particles in an indoor environment, so in order to disperse or reduce the accumulation of the infected droplets, modifying the configuration of ventilation systems may be an alternative solution to reduce particle transmissions (Zhao *et al.*, 2004; Kao *et al.*, 2009; Mateus *et al.*, 2016; Zhao *et al.*, 2021).

Due to the proliferation of harmful diseases, several experimental works investigating the particle flows inside buildings. Kao, Lai, Zhang, and Lu observed the movement of aerosol particle flows in two-zone rooms (Lu, and Howarth, 1996; Zhang and Chen, 2006; Lai *et al.*, 2008; Kao *et al.*, 2009), where the variables could indicate the efficiency of the forced ventilation system in terms of concentration and deposition, and the escape times of the particle. Some researchers studied particle movement in rooms with displacement and mixed ventilation systems (Zhao *et al.*, 2004; Gilani *et al.*, 2016; Mateus *et al.*, 2016), with the conclusion that the particle residence time could indicate of removal particle efficiency. The velocity and temperature are the focus for monitoring.

The Computational Fluid Dynamic (CFD) method is a tool that captures single-phase (air) and two-phase (particle-air) flows in both natural and forced convection systems (Promtong *et al.*, 2005; Promtong, 2006; Promtong, and Tekasakul, 2007; Tekasakul and Promtong, 2008; Promtong *et al.*, 2016; Promtong *et al.*, 2017; Promtong, 2018; Promtong *et al.*, 2018, Promtong *et al.*, 2020a, Promtong *et al.*, 2020b). Promtong used the CFD method to improve flow uniformity of smoked hot gas and temperature distribution in a smoking chamber (Promtong, and Tekasakul, 2007; Tekasakul, and Promtong, 2008), and Chang

and Zhao used the Lagrangian model to study the residence time of particles indoors (Zhao *et al.*, 2008; Chang *et al.*, 2013; Brusca *et al.*, 2016). By comparing the Lagrangian model and the Eulerian model, particle movements of various sized particles could be evaluated. Brusca studied the flow distribution of PM10 using Eulerian-Lagrangian 3D models (Brusca *et al.*, 2016), and the simulation work showed that the flow paths of PM10 could be tracked using the Eulerian system for continuous phase (the airflow) and using the Lagrangian system for Discrete Phase Model (DPM) (Particle Motion). Moreover, researchers have studied the influence of different components in a forced ventilation system to remove the particles, especially the air inlet angle and diffuser (Jurelionis *et al.*, 2015; Taheri *et al.*, 2021a, Taheri *et al.*, 2021b). Nevertheless, before the introduction of the CFD technique for particle flow studies, a validation study must be conducted (Promtong, and Tekasakul, 2007; Promtong *et al.*, 2016; Promtong *et al.*, 2017; Promtong *et al.*, 2020, Promtong *et al.*, 2020).

In this research, we will observe the fundamental physics of the particles in a forced ventilation system. The investigations will focus on how different particle sizes distributed inside a forced ventilation system are affected, and alternatives that could prevent infections and efficiently remove the particles from a building could be found and recommended. Overall, the objective of this research is to use the Computational Fluid Dynamics (CFD) technique;

1) to investigate the effect of ventilation configuration and the flow characteristics in a forced ventilation system,

2) to analyze the particle flow distribution and its flow paths in a forced ventilation system,

3) to study the influence of particle size, inlet velocity, and room configuration in relation to the flow, and to address these critical factors to improve particle removal efficiency.

The assessment by numerical accuracy of the forced ventilation system was performed as a mandatory task, and consequently, the validity of predictions related to the particle flow simulation could be further investigated. As well as, the variables considered, including the flow paths and the residence time, there will be a discussion regarding particle flow characteristics.

Fluid and Particle Flow Equations

A room's fluid flow can be described using Reynold Averaged Navier-Stroke (RAN) equations.

As the particle trajectory interest is evaluated, the Discrete Phase Model (DPM) is introduced for capturing the particle's flow characteristics, which should be considered if the fluid flow is turbulent. Hence the turbulence model is also required to calculate the average kinetic viscosity, as shown in the equation.

Continuity Equation

The mass equation expresses that the mass flow within a control volume is conserved (as shown in Equation (1)). Since the mass velocity is continuous, this partial differential equation is called the Continuity Equation. Occasionally, the first term can be omitted when the fluid flow is steady.

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial(\bar{u}_i)}{\partial x_i} = 0 \quad (1)$$

Momentum Equation

The momentum equation is described as in Equation (2), and the calculation concept is based on the momentum conservation within a control volume. Each direction's momentum equation has to be discretized and solved using matrix algebra to calculate the field velocity.

$$\frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right] \quad (2)$$

Where

$$\tau_{ij} = -\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \quad (3)$$

In Equations (1) and (2), ρ is the fluid density and $\bar{p}$ is the system pressure. $\bar{u}_i, \bar{u}_j$ is the average velocity components and $\overline{u'_i u'_j}$ is the velocity fluctuation. The x_i, x_j terms are the coordinate axis, and the gravitational acceleration vector and kinetic viscosity are represented by g_j and μ terms respectively. Equation (3) shows the Reynolds-stress term ($-\rho \overline{u'_i u'_j}$)

Moreover, the Boussinesq Approximation is increased as it calculates the difference between the properties of different fluids. This equation can be resolved using turbulent models (Launder, and Spalding, 1974; Chen, 1995; Chen *et al.*, 2006), where the turbulence model named $k - \varepsilon$ is used to calculate the airflow using the values shown in Equations (5) and Equation (6). and then bring to calculate the viscosity due to turbulence model in Equation (4).

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

k Equation:

$$\frac{\partial}{\partial x_i} (\rho k u_i) + \frac{\partial}{\partial t} (\rho k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon \quad (5)$$

ε Equation:

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) + \frac{\partial}{\partial t} (\rho \varepsilon) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (6)$$

Where

$$C_{2\varepsilon}^* = C_{2\varepsilon} + \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3}$$

And

$$\eta = S k / \varepsilon \quad S = (2 S_{ij} S_{ij})^{1/2}$$

The model constants are

$$C_\mu = 0.0845, C_{1\varepsilon} = 1.42, C_{2\varepsilon} = 1.68, \eta_0 = 4.38, \sigma_k = 0.7194, \sigma_\varepsilon = 0.7194, \beta = 0.0012$$

In Equation (6), the last term ($\rho \varepsilon$) is the destruction rate and P is the shear production buoyancy production term, which is produced by:

$$P = \mu_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (7)$$

The details of $k-\omega$ SST turbulent modelling are not given but it can be seen from the work of Menter (Menter, 1994). Recently, some turbulence models were evaluated by researchers the prediction capability using the open source CFD program (Suvanjumrat, 2017, Yuan *et al.*, 2017).

Discrete Phase Model (DPM) Modelling

The trajectory predictions of the discontinuous phase (particles) can be calculated using a special modelling method. According to the force equilibrium (Lagrangian reference frame), the particle's inertia can be equal with the forces acting on it; hence it can be written the relation as shown in Equation (8).

The remaining equations are used to calculate the required constant coefficients of the interfacial forces. Moreover, in this simulation study, the one-way approach can be adapted to model particle-primary fluid interactions when only the primary fluid flow affects the particle path. Additional information regarding the particle modelling can be found in the works of Zahari, Deen and Dickenson (Deen *et al.*, 2007; Dickenson and Sansalone, 2009; Zahari *et al.*, 2018).

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \quad (8)$$

$$F_D = \frac{18\mu}{\rho_p d_p^2} \frac{C_D Re}{24} \quad (9)$$

$$Re \equiv \frac{\rho d_p |u_p - u|}{\mu} \quad (10)$$

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (11)$$

Research Methodology

In this numerical study, the activities are divided into two main tasks. The validation study is the first critical task for completion when using the CFD technique, and the second task is the fundamental investigation of particle flow in different conditions.

Also, the key parameters are the velocity distribution and the particle residence time in the room (for each size), as these factors evaluate the ventilation system's performance most efficiently.

CFD Validation Study

Details of Room Models and Measuring Planes

The model room has the dimensions $0.8 \text{ m} \times 0.4 \text{ m} \times 0.4 \text{ m}$, which is related to the experimental investigation by Lai (Lai *et al.*, 2008). The room is

divided into two zones, and the inlet and outlet dimensions are both at $0.1 \text{ m} \times 0.04 \text{ m}$ (Figure 1(a)).

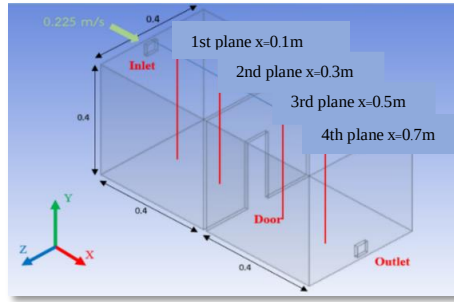
The door is located between the two rooms and is $0.1 \text{ m} \times 0.08 \text{ m} \times 0.24 \text{ m}$ in size. The figure shows four measuring planes, which are shown at $X = 0.1 \text{ m}$, 0.3 m , 0.5 m , and 0.7 m respectively. It should be noted that the measuring data on these planes were compared with the simulation results.

Meshing

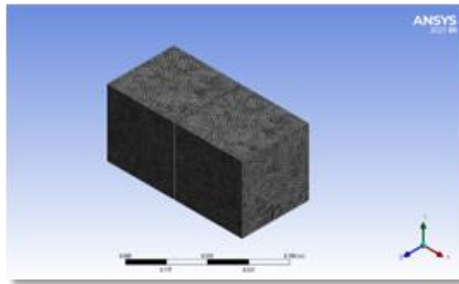
Figure 1(b) shows the model room after the meshing process and it shows the information regarding the effect of the elemental figures. It can be seen from Table 1 that the maximum grid sizes considered as mesh independence study are 0.05 m , 0.025 m , 0.015 m , and 0.010 m , respectively. After completing the benchmarking task, 155,375 was found to be the most suitable elemental number. By using this mesh number, the maximum error was less than 3.38% so this number size was used for further simulation investigations.

Table 1. Information of total mesh used for the model (mesh independence study)

Mesh Size (mm)	Total Meshes	Velocity (m/s)	Result Diff. (%)
0.050	7,774	0.059	47.92
0.025	42,972	0.094	17.76
0.015	155,375	0.110	3.38
0.010	445,031	0.114	0



(a) Geometry and Measuring Planes



(b) Flow Domain Mesh

Figure 1. Geometry of the Two-Zone Model Room and the Mesh of the Room

Results and Discussions

The first task is to study the airflow in a forced ventilation room, and this section will show a comparison between the simulations and experiments to address the capabilities of the numerical tool, especially in terms of prediction accuracy.

The simulation results were consistent with the experimental data shown in Figure 2, where the predicted results use different turbulence models, namely the RNG k-epsilon and SST k-omega were compared. After comparisons, the RNG k-epsilon model's prediction provided a better capture than the SST k-omega model. The maximum difference can be seen on the second plane at 0.355 m height located directly to the inlet.

Therefore, the simulation in the second plane switched to the SST k-omega model, resulting in discrepancies in this plane. Discrepancies may be reduced if considering more efficient turbulence closure, such as the Large Eddy Simulation (LES) model.

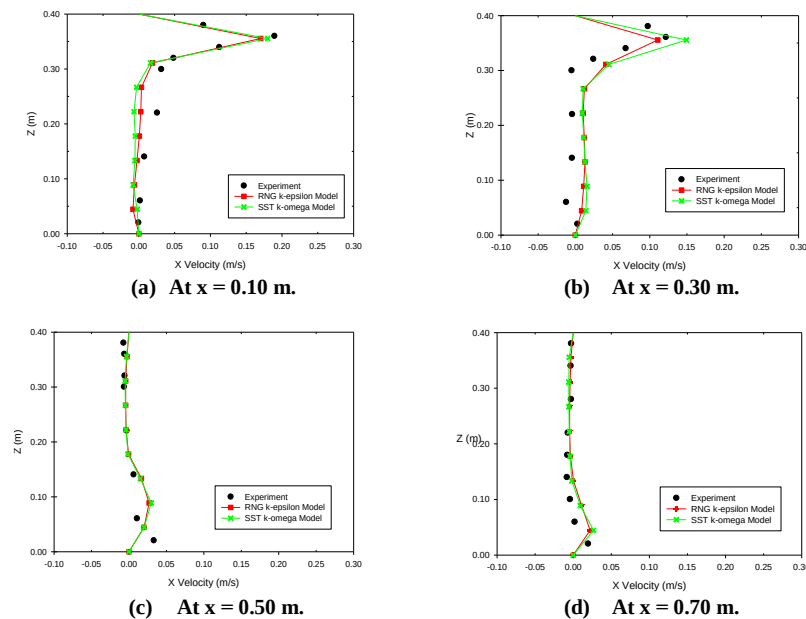


Figure 2. Comparison of the Velocity between the Measuring Values and the Predictions on four planes on the x-axis

Figure 3 shows the airflow characteristics in the model room in both Zone 1 and Zone 2. The air enters the room at a high velocity near the ceiling and changes direction after it hits the central wall (Zone 1). The highest velocities (0.20-0.26 m/s) are found in the middle plane in both zones.

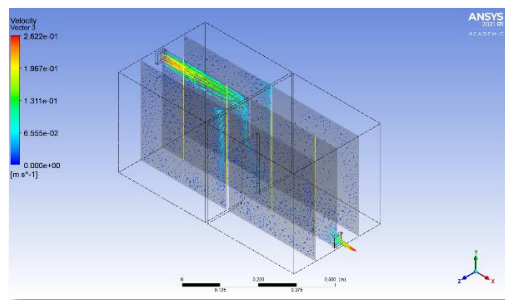


Figure 3. Flow Patterns Displaying the Middle Plane and the Right (Left) Planes

On average, the velocity in Zone 2 was lower than in Zone 1, especially on the right and left planes. The current ventilation, which is the displacement system, may enhance the efficiency of particle removal, and the following section will investigate the influence of different particle sizes.

Investigation of Particle Motions

The Discrete Phase Model (DPM) equation was used to track the particle movements during the flow calculation, and how the particle flow results are in line with the particle size, the velocity, and the ventilation system can be investigated and discussed. Generally speaking, nine particles of the same size ($1\ \mu\text{m}$, $5\ \mu\text{m}$, and $10\ \mu\text{m}$) were initially allocated evenly on the plane near the inlet wall and later flowed freely under the flow distributions. It should be noted that the main parameters to discuss are the Particle Residence Time (PRT) and the flow paths of each particle size.

Influence of the Particle Size

Figures 4(a), 4(b), and 4(c) show the Particle Residence Time (PRT) of the considered particle sizes ($1\ \mu\text{m}$ and $10\ \mu\text{m}$) and how they flow with the airstream through Zones 1 and 2. When comparing the path of the flows, the $1\ \mu\text{m}$ particles swirl in Zones 1 and 2 more than the $10\ \mu\text{m}$ particles. These larger particles were more likely to be affected by gravity and fall to the floor than the flow out of the room.

The simulation showed that $1\ \mu\text{m}$ particles could escape from the indoor room completely inside 13 min, but with the inlet velocity at 0.07 m/s the larger $10\ \mu\text{m}$ particles remained in the room for a longer period, finally escaping after 16 min.

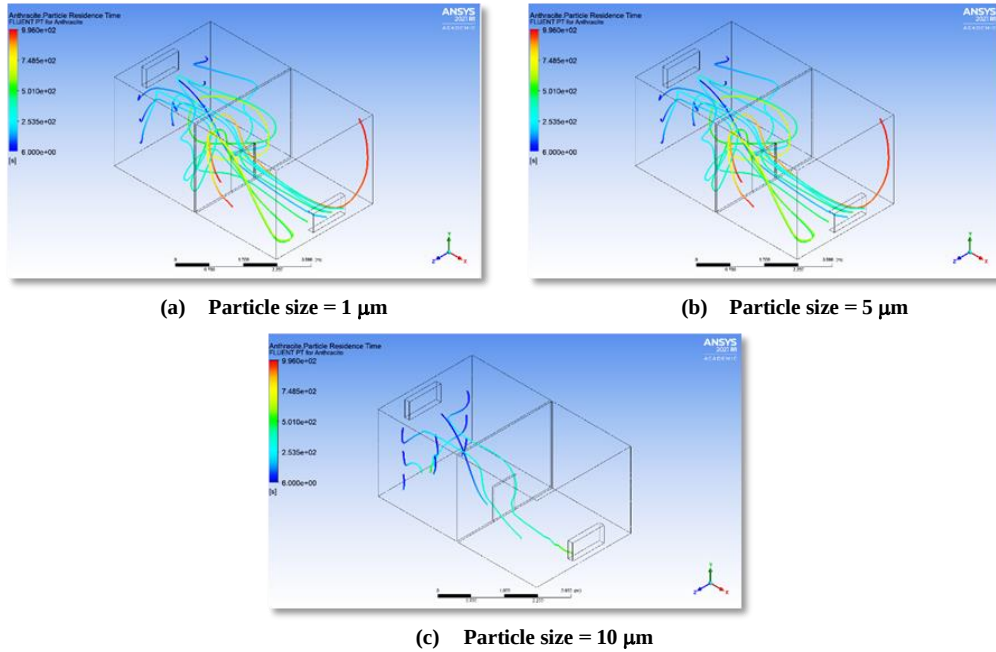


Figure 4. Particle Residence Time (PRT) of Particles Sized 1 μm (a), 5 μm (b), 10 μm (c) at a Speed of 0.07 m/s

Influence of the Inlet Velocity

Different particle sizes have different flow paths, and the velocity is also a critical factor in reducing the particle's time spent in the room. As shown in Table 2, the particles flowing with higher inlet velocity take a shorter time than the case of lower inlet velocity.

Table 2. The number of particles and the PRT of the particles before escaping the room

Particle Size (V_{inlet})	Z-1	Z-2	#Escaped (%)	Times escaped the room (mins.)
1 μm (0.07 m/s)	0	3	6 (66.67%)	3.2, 5.0, 6.3, 6.9, 7.8, 7.8
1 μm (0.14 m/s)	0	0	9 (100%)	2.5, 3.5, 3.6, 3.8, 3.9, 5.1, 5.5, 7.4, 7.6
10 μm (0.14 m/s)	3	0	6 (66.67%)	1.7, 3.2, 3.3, 3.7, 4.2, 4.3

Clearly, with the same inlet velocity (0.14 m/s), the particles of 1mm size travel from Zone 1 to Zone 2 quicker than the larger ones (10 mm). More details about the PRT that the particle sizes could escape the room under different inlet velocities are given in Table 2.

As shown in Figure 5, a higher inlet velocity (0.14 m/s) affects the flow path of the 1 μm flows, reducing the time taken before they were expunged. Interestingly, there were more circulations of small particles in Zone 2 than with lower velocities, but the conclusion is that the number of particles leaving the room is increased and the time taken less with higher velocity inputs.

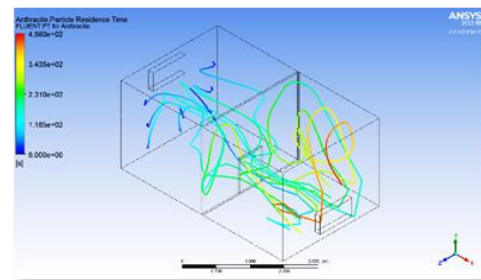


Figure 5. Particle Residence Time (PRT) of Particles Sized 1 μm at a Speed of 0.14 m/s

Influence of the Room Configuration (Additional Outlet)

Figure 6. shows the tracks of particle paths inside the modified room, which has two outlets. As the original room had one large outlet,

the displacement system allows particles to flow out more quickly. Modifications to the indoor structure regarding the inlet and outlet positions (mixed ventilation system) could influence particle motions.

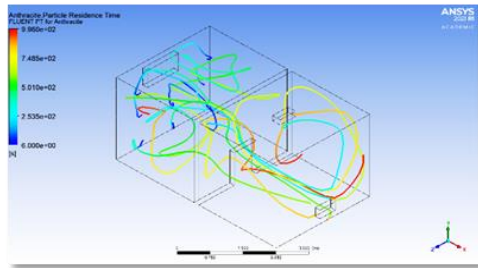


Figure 6. Particle Residence Time (PRT) of Particles Sized 1 μ m (having two outlets)

Noticeably, the new ventilation system influenced particle movement; consequently, the particles had flow paths differently. As a result, having an additional outlet could increase the PRT compared to the original room configuration.

Conclusions and Future Work

According to this numerical study, the fluid flow in a forced convection system (two-zone room), the particle flow characteristics in terms of Particle Residence Time (PRT), and the flow path for different sizes of particles have been successfully investigated. The following conclusions can be addressed:

The RNG k - ϵ model predicts the velocity profiles more accurately than the SST k - ω model.

The smaller particles (1-5 μ m) swirled before departing the forced ventilation room so had a longer PRT than the larger particles.

Most of the large particles (10 μ m) were deposited inside Zone 1 under these parameters. The higher inlet velocity decreased the PRT for all particle sizes.

An additional outlet provided more circulation into Zone 2 so the PRT of the modified room (mixed ventilation) was higher than the original configuration (displacement system).

In future work, more understanding regarding the flow of small particles will be utilized when simulating the forced ventilation system. Also, modifications in the displacement system could be helpful for reducing transmissions of infected particles.

Acknowledgements

Many thanks to the Mechanical Engineering Department, Faculty of Engineering, Mahidol University, for allowing us to use the laboratory.

Thanks to CADFEM SEA Pte. Ltd. - A certified Elite Channel Partner of ANSYS - for providing support and access to the CFD program named ANSYS Fluent R21.

Also, many thanks to Robert McEvoy, from the Faculty of ICT at Mahidol University, for editing and proofreading this original manuscript.

References

- Brusca, S., Famoso, F., Lanzafame, R., Mauro, S., Messina, M., and Strano, S. (2016). PM10 Dispersion Modeling by Means of CFD 3D and Eulerian-Lagrangian Models: Analysis and Comparison with Experiments. *Energy Procedia.*, 101:329-336.
- Chang, T.J., Kao, H.M., and Yam, R.S.W. (2013). Lagrangian modeling of the particle residence time in indoor environment. *Building and Environment.*, 62:55-62.
- Chen, F., Yu, S.C. M., and Lai, A.C.K. (2006). Modeling particle distribution and deposition in indoor environments with a new drift-flux model. *Atmospheric Environment.*, 40(2):357-367.
- Chen, Q. (1995). Comparison of different k - ϵ models for indoor air flow computations. *Numerical Heat Transfer, Part B: Fundamentals.*, 28(3):353-369.
- Deen, N.G., Van Sint Annaland, M., Van Der Hoef, M.A., and Kuipers, J.a.M. (2007). Review of discrete particle modeling of fluidized beds. *Chemical Eng. Sci.*, 62(1-2):28-44.
- Dickenson, J.A. and Sansalone, J.J. (2009). Discrete phase model representation of particulate matter (pm) for simulating |pm separation by hydrodynamic unit operations. *Environmental Sci. and Technol.*, 43(21):8,220-8,226.
- Gilani, S., Montazeri, H., and Blocken, B. (2016). CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. *Building and Environment.*, 95:299-313.
- Jurelionis, A., Gagytė, L., Prasauskas, T., Čiužas, D., Krugly, E., Šeduikytė, L., and Martuzevičius, D. (2015). The impact of the air distribution method in ventilated rooms on the aerosol particle dispersion and removal: The experimental approach. *Energy and Buildings.*, 86:305-313.
- Kao, H.M., Chang, T.J., Hsieh, Y.F., Wang, C.H., and Hsieh, C.I. (2009). Comparison of airflow and particulate matter transport in multi-room buildings for different natural ventilation patterns. *Energy and Buildings.*, 41(9):966-974.
- Lab, B. (2021) Particle Air Cleaning and Sick Building Syndrome Symptoms. See. Available from: <https://iaqscience.lbl.gov/air-sick> Accessed date: May 05, 2021.
- Lai, A.C.K., Wang, K., and Chen, F.Z. (2008). Experimental and numerical study on particle distribution in a two-zone chamber. *Atmos Environ* (1994)., 42(8):1,717-1,726.
- Launder, B.E. and Spalding, D.B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering.*, 3(2):269-289.
- Lu, W. and Howarth, A.T. (1996). Numerical Analysis of Indoor Aerosol Particle Deposition and Distribution in Two-Zone Ventilation System. *Building and Environment.*, 31(1):41-50.

- Mateus, N.M., Simões, G.N., Lúcio, C., and Da Graça, G.C. (2016). Comparison of measured and simulated performance of natural displacement ventilation systems for classrooms. *Energy and Buildings*, 133:185-196.
- Menter, F.R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.*, 32(8):1,598-1,605.
- Promptong, M. (2006). CFD study of flow in rubber smoking room and modification to improve uniformity of velocity and temperature distributions. Prince of Songkla University., 27(11):2,113-2,121.
- Promptong, M. (2018). Development of mechanistic approach for wall heat flux partitioning in subcooled boiling flows. RMIT University., 174.
- Promptong, M., Cheung, S., and Tu, J. (2016). Numerical modelling of subcooled boiling flow based on mechanistic approach: A validation study using wet steam (IAPWS) as working fluid properties. *Lecture notes in engineering and computer science*, 1,067-1,075.
- Promptong, M., Cheung, S., Yeoh, G., Vahaji, S., and Tu, J. (2017). CFD study of subcooled boiling flow at elevated pressure using a mechanistic wall heat partitioning model. *World Acad Sci. Eng. Technol. Int. J. Mech. Aerosp Ind Mechatron Manuf Eng.*, 10(4):175,748.
- Promptong, M., Cheung, S., Yeoh, G., Vahaji, S., and Tu, J. (2018). CFD investigation of sub-cooled boiling flow using a mechanistic wall heat partitioning approach with Wet-Steam properties. *The J. of Com. Multiphase Flows*, 10(4):239-258.
- Promptong, M., Kasemjirapatara, C., Srithep, P., Masoodi, Y., Namchantha, S., Priyadumkol, J., and Suvanjumrat, C. (2020a). Investigation of aerodynamic performance of four potential airfoils for a formula sae car: a 2d validation study. In: *The 34th Conference of the Mechanical Engineering Network of Thailand*, 15-17 July 2020, Prachuap Khiri Khan, Thailand, p. 495-508.
- Promptong, M., Khunsri, K., Teachapanitvittaya, K., Trakulkumlu, T., Watechagit, S., and Suvanjumrat, C. (2020b). Experimental and numerical investigations into the natural convection of hot gas in a vertical smoking oven: a validation study. In *The 34th Conference of the Mechanical Engineering Network of Thailand*, 15-17 July 2020, Prachuap Khiri Khan, Thailand, p. 484-494.
- Promptong, M., Kirirat, P., Taweekun, J., and Tekasakul, P. (2005). CFD and Experimental Study of the Temperature Distribution in a Present Rubber Smoking Room In *Proceedings of The 19th Conference of Mechanical Engineering Network of Thailand*.
- Promptong, M. and Tekasakul, P. (2007). CFD study of flow in natural rubber smoking-room: I. Validation with the present smoking-room. *Applied Thermal Engineering*, 27(11-12):2,113-2,121.
- Ren, J., Wang, Y., Liu, Q., and Liu, Y. (2021). Numerical Study of Three Ventilation Strategies in a prefabricated COVID-19 inpatient ward. *Build Environ.*, 188:107,467.
- Suvanjumrat, C. (2017). Comparison of Turbulence Models for Flow Past NACA0015 Airfoil Using OpenFOAM. *Engineering J.*, 21(3):207-221.
- Taheri, M., Zolfaghari, S.A., Afzalian, M., and Hassanzadeh, H. (2021a). The influence of air inlet angle in swirl diffusers of UFAD system on distribution and deposition of indoor particles. *Building and Environment*, 191.
- Taheri, M., Zolfaghari, S.A., Hassanzadeh, H., and Salmanzadeh, M. (2021b). Numerical investigation of the effects of fan-coil airflow direction on distribution and deposition of indoor pollutant particles. *J. of Building Eng.*, 33.
- Tekasakul, P. and Promptong, M. (2008). Energy efficiency enhancement of natural rubber smoking process by flow improvement using a CFD technique. *Applied Energy*, 85(9):878-895.
- Vuorinen, V., Aarnio, M., Alava, M., Alopaeus, V., Atanasova, N., Auvinen, M., Balasubramanian, N., Bordbar, H., Erasto, P., Grande, R., Hayward, N., Hellsten, A., Hostikka, S., Hokkanen, J., Kaario, O., Karvinen, A., Kivistö, I., Korhonen, M., Kosonen, R., Kuusela, J., Lestinen, S., Laurila, E., Nieminen, H.J., Peltonen, P., Pokki, J., Puisto, A., Raback, P., Salmenjoki, H., Sironen, T., and Osterberg, M. (2020). Modelling aerosol transport and virus exposure with numerical simulations in relation to SARS-CoV-2 transmission by inhalation indoors. *Saf Sci.*, 130:104,866.
- W.H.O. (2020) Coronavirus disease (COVID-19): How is it transmitted? World Health Organization [WHO], See. Available from: <https://www.who.int/news-room/q-a-detail/coronavirus-disease-covid-19-how-is-it-transmitted>. Accessed date: May 05, 2021.
- W.H.O. (2021) Air pollution. World Health Organization [WHO], See. Available from: https://www.who.int/health-topics/air-pollution#tab=tab_1. Accessed date: May 01, 2021.
- Yuan, H.L., Kekina, P., Suvanjumrat, C., Agarwal, R.K., Tandon, P., and Wang, E.X. (2017). A Comparative Study on Turbulence Models for Simulation of Flow Past NACA 0015 Airfoil Using OpenFOAM. *MATEC Web. of Conferences*, p. 95.
- Zahari, N.M., Zawawi, M.H., Sidek, L.M., Mohamad, D., Itam, Z., Ramli, M.Z., Syamsir, A., Abas, A., and Rashid, M. (2018) Introduction of discrete phase model (DPM) in fluid flow: A review. In: *Green Design and Manufacture: Advanced And Emerging*, 2030(1):020,234.
- Zhang, Z. and Chen, Q. (2006). Experimental measurements and numerical simulations of particle transport and distribution in ventilated rooms. *Atmospheric Environment*, 40(18):3,396-3,408.
- Zhao, B., Yang, C., Yang, X., and Liu, S. (2008). Particle dispersion and deposition in ventilated rooms: Testing and evaluation of different Eulerian and Lagrangian models. *Building and Environment*, 43(4):388-397.
- Zhao, B., Zhang, Y., Li, X., Yang, X., and Huang, D. (2004). Comparison of indoor aerosol particle concentration and deposition in different ventilated rooms by numerical method. *Building and Environment*, 39(1):1-8.
- Zhao, W., Kilpeläinen, S., Kosonen, R., Jokisalo, J., Lestinen, S., and Mustakallio, P. (2021). Thermal environment and ventilation efficiency in a simulated office room with personalized micro-environment and fully mixed ventilation systems. *Building and Environment*, 188.