Penerbit Akademia Baru

CFD Letters

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


# Evaluation of Air Flow Pattern for Conceptual Design of Automotive Painting Line Using Computational Fluid Dynamic (CFD) for Better Dust Particle Reduction

Open Access

Muhammad Hafizan Yosri[1], Pauziah Muhamad[1,*], Norfazrina Hayati Mohd Yatim[2]

[1] Intelligent Dynamics & System Research Lab, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Malaysia
[2] Department of Mechanical Engineering, Kulliyyah of Engineering, International Islamic University Malaysia, 50728 Kuala Lumpur, Malaysia

**ARTICLE INFO**

***Article history:***
Received 11 October 2018
Received in revised form 27 December 2018
Accepted 1 January 2019
Available online 9 February 2019

**ABSTRACT**

In the painting production process, repairing of paint work defects by running the part through repeat process, together with the essential requirement quality control routines, account for a very large proportion of the operating costs .The dust and fibre defects is ranged between 40% and 50% and found to be the highest rejection from one of the local industry painting line. Hence that, this research is focused to study the effectiveness of simulating this painting line using Computational Fluid Dynamic (CFD) method to identify the air flow and turbulence pattern which may help to understand the particle concentration and movement in the painting line. Six mechanical designs of ventilation system was proposed in order to study the particle movement and concentration to this automotive painting line. From here, the best design which suite the objective of minimizing the particle concentration and its dissipation into the painted part are justified. Hence, improvement action such as layout arrangement and mechanical design are factors involved to overcome and minimize the foreign particle from fall down into the parts during the painting process. The result from this study may also be a benchmarking for future design of new automotive painting line.

***Keywords:***
Computational Fluid Dynamic, Turbulence Models RNG *k*–*ε* model, Particle concentration, Automotive Painting Line.

**Copyright © 2019 PENERBIT AKADEMIA BARU - All rights reserved**

## 1. Introduction

Dust and fibre have been identified among the highest contributor for the defect in automotive painting line with a range from 40% to 50% of total defect breakdown [1]. Eventually, those defects will effect on both visual appearance and also performance of the parts. In addition, the significance of controlling dust in the production line is crucial in order to maintain the quality of the product, part performance yield and the effect on workers' health [2]. In order to reduce the airborne contamination, many manufacturer in electronic and food industries are using the clean room method to control the dust particle by using the air distribution system that related to Computational

* *Corresponding author.*
*E-mail address: pauziah.kl@utm.my (Pauziah Muhamad)*

Fluid Dynamic (CFD) [3]. By doing so, the air flow pattern and turbulence that affects the dust and fiber concentration in painting line can be identified using CFD which is one of the primary methods used to assess indoor airflow [4].

CFD has widely been applied to various engineering applications such as automobile and aircraft design, weather science, civil engineering, and oceanography. Srebic *et al.,* [5] used CFD to predict contaminant dispersion around human occupants. Their study revealed that the optimum air speed, the speed resulting in the lowest particle counts was in the range of 0.35~0.55 m/s. These findings were also aligned and supported with the study conducted by Jatuporn *et al.,* [6] especially at the effective Fan Filter Unit (FFU) speed in range of 0.35~0.55 m/s to block and purging the dust from enter the clean room. Rouaud and Havet [7] have applied CFD to solve the contamination on food processing in clean room. In addition, Noh *et al.,* [8] simulated the particle movement in LCD manufacturing clean room. By adopting the particle tracking techniques, the movement of air and other particulate matter can be predicted [9]. Therefore, CFD is proven to be able to identify areas of stagnation and recirculation in complex situation [10].

Air distribution in an enclosed environment can be driven by different forces, such as natural wind, mechanical fan or thermal buoyancy. The combination of these flow mechanisms (forced, natural, and mixed convection) creates complex indoor airflow characteristics with impingement, separation, circulation, reattachment, vortices and buoyancy (Figure 1).

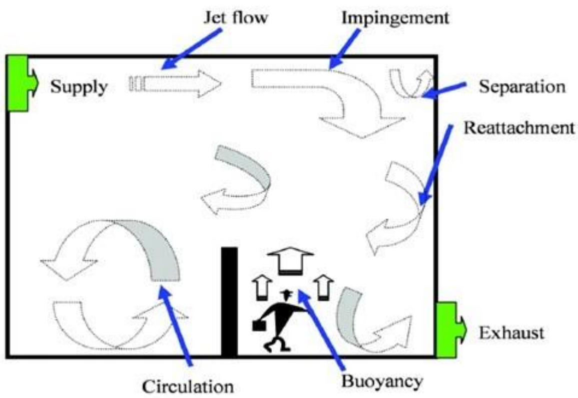


**Fig. 1.** Typical flow characteristics in an enclosed environment with various flow mechanisms.

Whyte *et al.,* [11] carried out a series of experiments and bacteriological evaluations in an Operation Room equipped with Laminar Air Flow systems. They have found that the down-flow (horizontal) system is bacteriologically superior to the cross-flow (vertical) when conventional clothing is used. Since the existing ventilation system in the automotive painting line has already established the horizontal laminar air flow system, the research for particle movement can be narrowed down by focusing on the horizontal ventilation system with focusing on the interior air circulation and exhaust system.

## 2. Methodology

The k-ε model family is the most popular turbulence model and has the largest number of variants. The “standard” *k-ε* model developed by Launder and Spalding [12] is one of the most prevalent models for indoor airflow simulation due to its simple format, robust performance, and wide validations. The turbulent eddy viscosity, $v_t$, is calculated in the k-ε model as follows:

$$v_t = C_\mu \frac{k^2}{\varepsilon} \tag{1}$$

where k is the turbulence kinetic energy, ε is the dissipation rate of turbulence energy, and Cμ = 0.09 is an empirical constant. To apply the model for low Reynolds number flows, such as near-wall flows, wall functions [12] are usually used to connect the outer-wall free stream and the near-wall flow. The standard k-ε model with wall functions was commonly used and produces acceptable results [13]. However, the model is facing difficulty in dealing with certain special room situations such as high buoyancy effect and large temperature gradient.

Yakhot *et al.,* [14] has improved the RNG k-ε model by providing similar but slightly better results compared to existing Standard k-ε model. Although the physics is treated in a simplistic manner, the model works surprisingly well in many types of flows [15]. Chen [16] previously evaluated the performance of five k–ε models for the simulation of forced convection, natural convection, and mixed convection room airflow. The results from the RNG k–ε model were slightly better than those from the standard k–ε model, and the other models. This model was developed using renormalization group theory and can be written as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\epsilon \tag{2}$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}P_k - C_{2\epsilon}^{*}\rho\frac{\epsilon^2}{k} \tag{3}$$

where

$$C_{2\epsilon}^{*} = C_{2\epsilon} + \frac{C_\mu \eta^3(1-\eta/\eta_o)}{1+\beta\eta^3} \tag{4}$$

with

$$\eta = Sk/\epsilon \tag{5}$$

and

$$S = \left(2S_{ij}S_{ij}\right)^{1/2} \tag{6}$$

*k* is a turbulence kinetic energy per unit mass and e is turbulence dissipation rate, more details and all the constants are given by Yakhnot *et al.,* [17].

### *2.1. Mechanical Design Proposal*

A total of six mechanical designs of ventilation system in automotive painting line were proposed in order to study the dust particle movement and concentration. The mechanical design of the ventilation system is focusing on the location of the outlet, size and orientation of the exhaust (outlet system) in the painting line system (Figure 2).

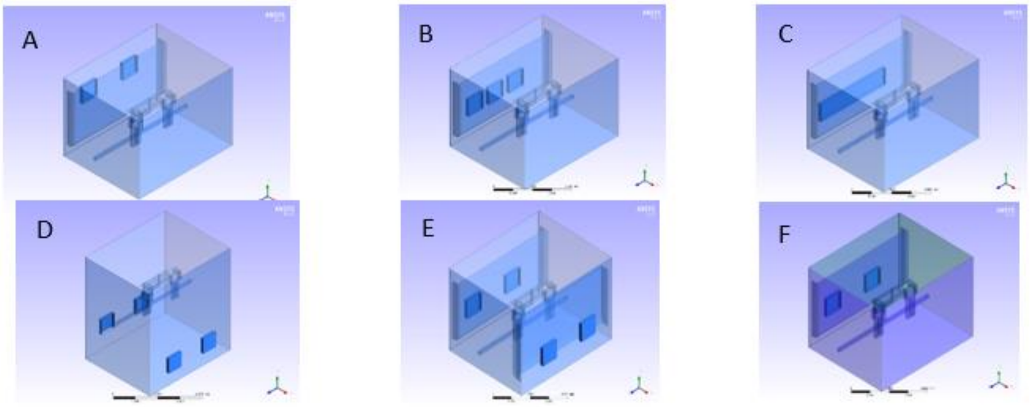


**Fig. 2.** Mechanical design proposal for air ventilation system in automotive painting line: (A) Change the position of exhaust, (B) Additional exhaust, (C) New exhaust design - Horizontal design, (D) Air Curtain - Down draft system, (E) Air Curtain - Add position (left/ right) and (F) Initial/current condition

### *2.2. Parameter at Boundary Conditions*

The inlet conditions set at a uniform velocity of 0.455 m/s and the inlet turbulence intensity is 4%. These conditions are consistent with Nielsen's test case [18].The outlet condition was a constant pressure of zero. The velocity was set to zero at the room walls and outlet walls to impose a no-slip condition. The left and right walls and the ceiling were assumed to be adiabatic. The commercial software FLUENT 15.0.7 was used to perform the numerical simulations and a SIMPLE algorithm were selected to run the simulation. A second-order upwind scheme was used for discretizing the convection terms. Convergence criteria for the continuity and energy equations were set to be 0.02. The particle movement in the ventilated system of painting line was simulated using anthracite particle by applying Lagrangian-based discrete random walk (DRW) model and the particle size was set at 1e-6. The dispersion of particles was accounted with the stochastic discrete-particle approach. The mesh structure with an average of 1800000 nodes from 6 different design was chosen.

### *2.3 Validation and Convergence Study*

Simulation convergence should be satisfied the following factor. All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance or the solution no longer changes with subsequent iterations. In addition, overall mass, momentum, energy, and scalar balances must be achieved. As for benchmarking for the lowest particle concentration (mechanical model E), the convergence was archived at 317 steps with the momentum value was set at 0.03 (as shown in Figure 3). At this stage, necessary action has been planned to run the experiment in a small scale model of automotive painting line.

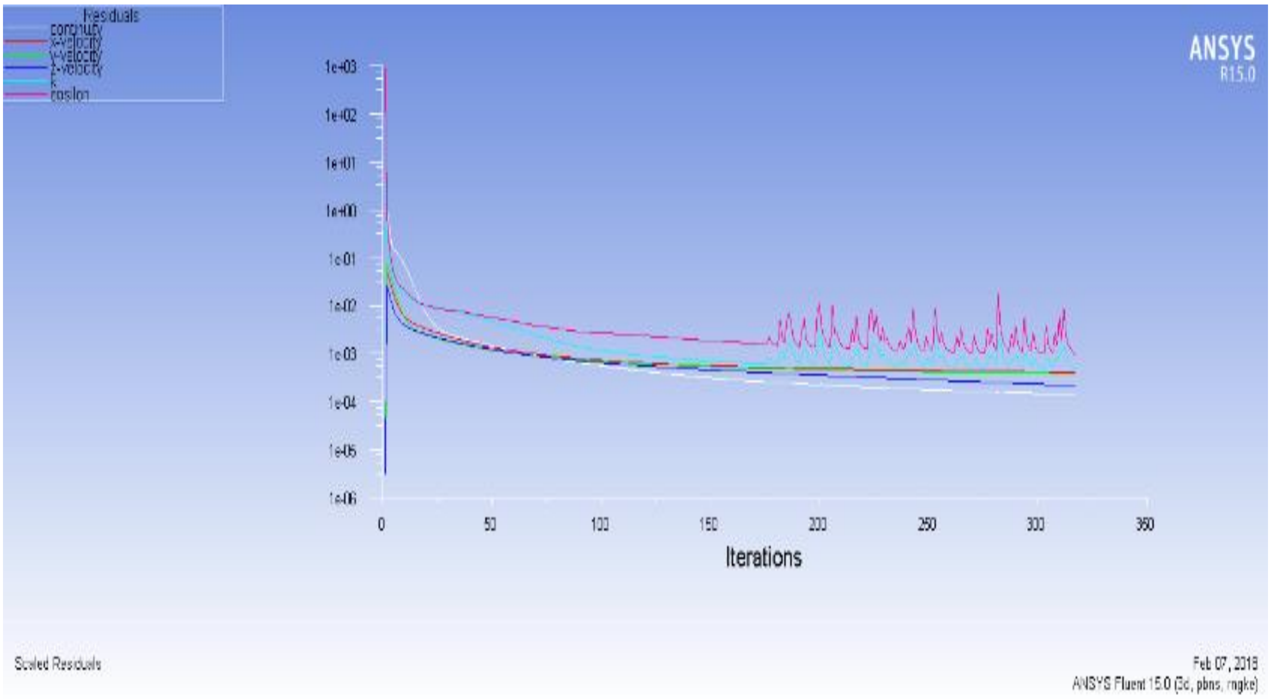


**Fig. 3.** Convergence for simulation at mechanical model (E) Air Curtain - Add position (left/ right)

## 3. Results and discussion

The results from six mechanical models of automotive painting line were evaluated based on the velocity vector and particle mass concentration. The particle mass concentration of anthracite particle was monitored from the particle dispersion during ANSYS Fluent Simulation. The summary of particle distribution in horizontal plane for each model can be referred from the Figure 4.

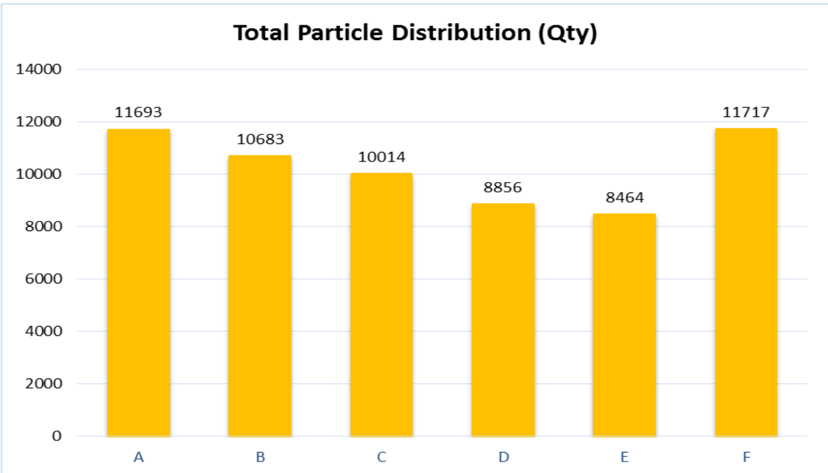


**Fig. 4.** Total particle distribution at horizontal plane

The ventilation system was tested by expanding the size, relocation and installation of additional outlet system to ensure particle distribution can be reduced. The result shows by changing the location and design of exhaust system, the particle distribution in the ventilated system of automotive painting line can be improved. From the result, the F model which is the initial design of automotive painting line provide the highest particle distribution from the other models. From six conceptual designs that have been simulated, the E model shows the prevalent result compared to others. Even though all presented model shows significant improvement compared to the existing model, the E model shows the best result by judging at the particle distribution in the ventilated system. In overall, model E provides in total 8464 particles recorded in horizontal plane compared to current painting line model F with 11717 particles.

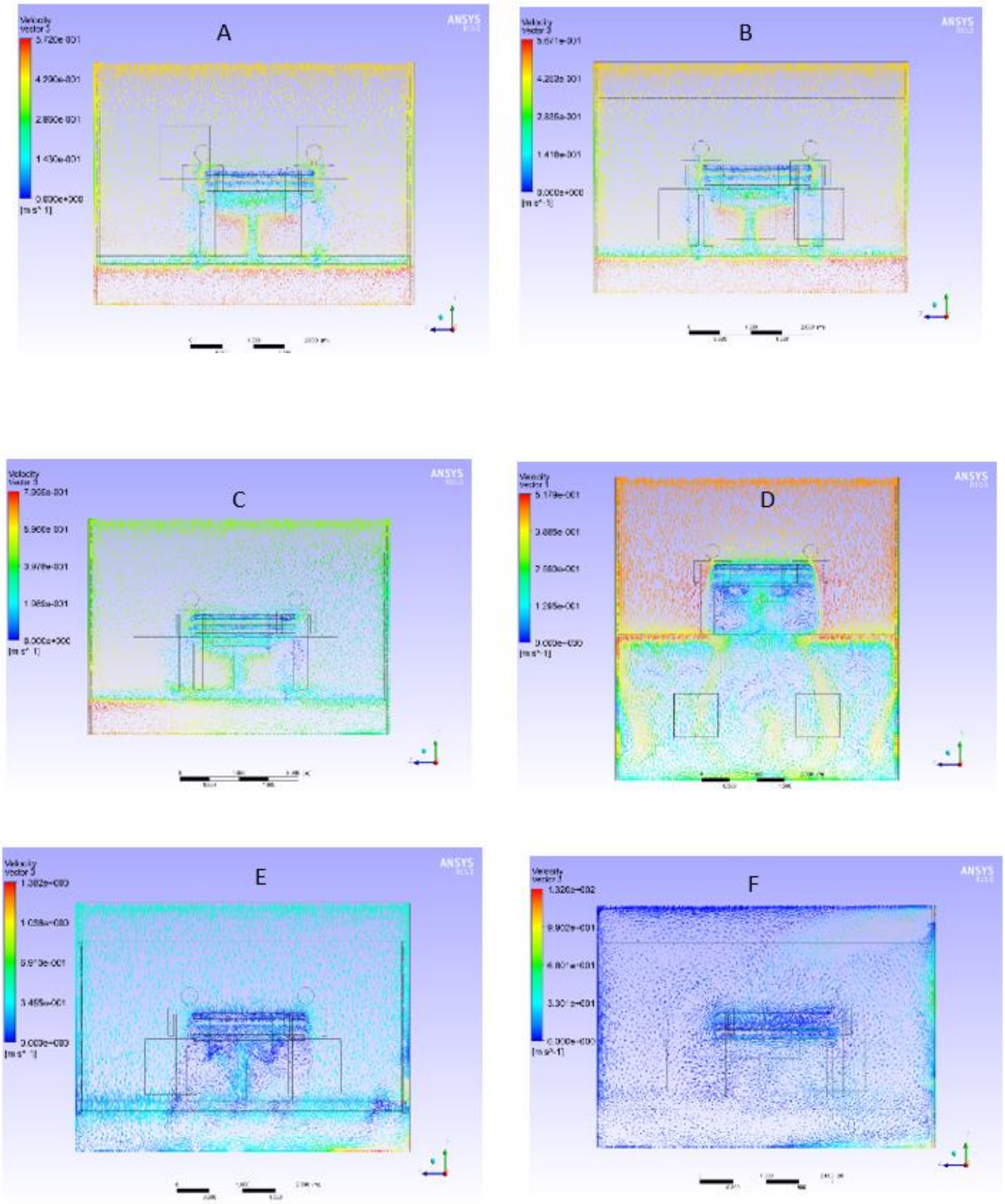


**Fig. 5.** Velocity vector for air ventilation system in automotive painting line: (A) Change the position of exhaust, (B) Additional exhaust, (C) New exhaust design - Horizontal design, (D) Air Curtain - Down draft system, (E) Air Curtain - Add position (left/ right) and (F) Initial/current condition.

The main factors that affect the particle distribution is the velocity vector and the mechanical design of ventilation system (as shown in Figure 5). By having higher velocity vector, the idea is to swipe away and reduce the particle concentration from the ventilation system. However, an efficient ventilation is not only a matter of increasing airflow velocity. For example, simulation for model A, B and D provide among highest velocity vector compared to other models with the average velocity speed is 0.3886 m/s. However, the particle concentration is still higher compared to model E. This situation might be contributed from inappropriate airflow rates which result in flow pattern transition from laminar to less efficient turbulent mixing. This shows that by increasing the air exchange rate alone will not always guarantee sufficient control of the particle movement within the ventilation system. In this situation, airborne particle was carried out by air flow towards painted area and settle down into the painted part. For the other models with high velocity vector, except models E and F, the air flows create a minor turbulence circulation around the painted area which contribute to the particle concentration and accumulation around painted surface. Even with high velocity vector around the horizontal plane, the outcome of particle concentration will become uncertain when improper and ineffective ventilation design is not considered. For example, the model E has an average velocity vector is around 0.346 m/s with consistent laminar air flow provide less particle concentration and reduced the turbulence effect around the painted parts.

On other hand, the F model (current system) provides the lowest velocity vector through the horizontal plane. Due to low velocity of the air flow rate, sedimentation velocity of emitted particles appeared to be slightly higher. In other words, the settling velocity was found to be higher at low velocity. Despite the velocity vector from this model appear to be much lower than models, the results show that accumulation of the particles in the system is the highest among all six tested models. In overall, there is 27.8% reduction of particle concentration from the improved model E compared to existing model F. The reduction was contributed by improved laminar air flow with higher velocity vector and less particle concentration around the painted surface.

## 4. Conclusions

Airflow and particle distribution in automotive painting line have been predicted numerically to investigate the effects of various mechanical design of ventilation system on the flow characteristics. For this purpose, the RNG k–ε model is preferred because of its slightly better prediction of air flow and works surprisingly well with many types of flows. Different turbulence models may produce different solutions under the same boundary conditions. Selection of multiple mechanical design of ventilation system particularly on the exhaust system to flush and sweep off the particle in the system depend on many factors including the location of the outlet, size and machine capacity. In this simulation, the painting line parameter have been fixed to be constant to measure the effectiveness of the proposed mechanical design. Positive outcome depends to the effectiveness of the ventilation system to sweep and reduce the particle concentration in the painting line and minimize the particle movement towards the painted part by understanding the air velocity vector. The results confirmed that by adding more exhaust fan at opposite direction can improve the air flow circulation and reduce the particle concentration in the painting line. Future works such as experiment on small scale of automotive painting line shall be conducted in order to verify the result obtained from the numerical simulation done using ANSYS Fluent.

## References

[1] Combat Coating (M) Sdn Bhd, Outgoing Quality Inspection Passrate Report, 2015~2016.

[2] Bascom, Rebecca, Jana Kesavanathan, and David L. Swift. "Human susceptibility to indoor contaminants." *Occupational Medicine (Philadelphia, Pa.)* 10, no. 1 (1995): 119-132.

[3] Holbrook, Daniel. "Controlling contamination: the origins of clean room technology." *History and Technology* 25, no. 3 (2009): 173-191.

[4] Havet, M., and D. Blay. "Numerical study of coupled convective, radiative and conductive heat transfer in a large enclosure. Application to radiant heating, Roomvent'94." In *Fourth international conference on air distribution in rooms,* pp. 15-17. 1994.

[5] Srebric, Jelena, Vladimir Vukovic, Guoqing He, and Xudong Yang. "CFD boundary conditions for contaminant dispersion, heat transfer and airflow simulations around human occupants in indoor environments." *Building and Environment* 43, no. 3 (2008): 294-303.

[6] Thongsri, Jatuporn, and Monsak Pimsarn. "Optimum airflow to reduce particle contamination inside welding automation machine of hard disk drive production line." *International journal of precision engineering and manufacturing* 16, no. 3 (2015): 509-515.

[7] Rouaud, O., and Michel Havet. "Numerical investigation on the efficiency of transient contaminant removal from a food processing clean room using ventilation effectiveness concepts." *Journal of Food Engineering* 68, no. 2 (2005): 163-174.

[8] Noh, Kwang-Chul, Hyuk-Soon Kim, and Myung-Do Oh. "Study on contamination control in a minienvironment inside clean room for yield enhancement based on particle concentration measurement and airflow CFD simulation." *Building and Environment* 45, no. 4 (2010): 825-831.

[9] Ahmad, Rosmaini, Shahrul Kamaruddin, Zahid A. Khan, Mohzani Mokthar, and Indra Putra Almanar. "Implementation of dust control system using management and planning tools (MPT) A case study." *Management of Environmental Quality: An International Journal* 17, no. 4 (2006): 390-408.

[10] Hayakawa, Ichiya, Shuji Fujii, and Kwang Young Kim. "Design theory for a laminar flow-type clean room and image processing on remotely detected particulates." *Aerosol science and technology* 7, no. 1 (1987): 47-56.

[11] Whyte, W., B. H. Shaw, and R. Barnes. "A bacteriological evaluation of laminar-flow systems for orthopaedic surgery." *Epidemiology & Infection* 71, no. 3 (1973): 559-564.

[12] Launder, B. E., and D. B. Spalding. "The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanical Engineering, 3, 269-289." (1974).

[13] Zhang, Zhao, Wei Zhang, Zhiqiang John Zhai, and Qingyan Yan Chen. "Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2—Comparison with experimental data from literature." *Hvac&R Research* 13, no. 6 (2007): 871-886.

[14] Yakhot, Victor, and Steven A. Orszag. "Renormalization group analysis of turbulence. I. Basic theory." *Journal of scientific computing* 1, no. 1 (1986): 3-51.

[15] L. Davidson, P.V. Nielsen. "Large eddy simulations of the flow in a three dimensional ventilated room" In *Proceedings of the 5th International Conference on Air Distribution in Rooms Roomvent*'96, Yokohama (1996): 161–168.

[16] Chen, Q. "Comparison of different k-ε models for indoor air flow computations." *Numerical Heat Transfer, Part B Fundamentals* 28, no. 3 (1995): 353-369.

[17] Yakhot V, Orszag SA, Thangam S, Gatski TB, Speziale CG. "Development of turbulence models for shear flows by a double expansion technique." *Physics of Fluids A: Fluid Dynamics* 4, no. 7 (1992): 1510-1520.

[18] Nielsen, Peter Vilhelm. "Specification of a two-dimensional test case:(iea)." (1990).