ADVANCES IN AGRICULTURAL AND FOOD RESEARCH JOURNAL

MALAYSIAN SOCIETY OF AGRICULTURAL AND FOOD ENGINEERS
Est. 1982

HH PUBLISHER

Original Research Article

Plant Factory Airflow Distribution Analysis with Different Inlet Configuration

Arina Mohd Noh*, Muhd Akhtar Mohd Tahir, Khairul Anuar Shafie

Engineering Research Centre, Malaysia Agricultural Research & Development Institute, Serdang, Selangor 43400, Malaysia

*Corresponding author: Mohd Noh, A, Engineering Research Centre, Malaysia Agricultural Research & Development Institute, Serdang, Selangor 43400, Malaysia; arina@mardi.gov.my

Abstract: Airflow is important in plant factories as it is responsible for the air exchange inside the structure to create desired growing conditions for plants. A uniform airflow distribution enhances photosynthesis and the transpiration process of the plants. In this study, computational fluid dynamics (CFD) analysis was used to analyse the airflow distribution inside a commercial scale plant factory developed by MARDI. CFD plays an important role in designing and optimisation of control environment structure in the agriculture industry. Many studies have proved that the CFD technique is able to predict the internal climate of the plant factory in the designing stage before the actual plant was built. This study was conducted to analyse the airflow characteristics in a plant factory with different inlet and outlet locations. The study also analyses the effect of different inlet location to the overall temperature distribution inside the plant factory. Validation of the developed CFD model was carried out by comparing simulation results with experimental data. The validation result showed an acceptable percentage error between simulated and actual data. The validated CFD model was then used to analyse different inlet locations that can produce more uniform airflow and temperature distribution inside the plant factory. From the simulation results, it shows that the new inlet location was able to produce more uniform airflow and temperature distribution as compared to existing inlet location.

Keywords: CFD simulation, plant factory, airflow distribution, uniform

Received: 16th March 2021

Received in Revised Form: 14th July 2021

Accepted: 3rd August 2021

Available Online: 24th August 2021

Citation: Mohd Noh, A, Mohd Tahir, M.A, & Shafie, K. A, Plant factory airflow distribution analysis with different inlet configuration. *Adv Agri Food Res J* 2021; 2(2): a0000242. https://doi.org/10.36877/aafrj.a0000242

1. Introduction

It is important to understand the internal airflow in an indoor plant factory in detail to effectively deliver conditioned air to cultivation beds in order to maintain climate uniformity and promote adequate air movement around crops. A study by Kozai and Takagaki (2015) has proved that air movement plays an important role in aerodynamics at leaf surfaces. It affects the gas, heat and water exchange of plants and thus affects plants transpiration and photosynthetic rates. The study proved that increasing airflow speeds for both vertical and horizontal directions from 0.01 to 0.30 ms^{-1} around crops can significantly enhance the plants transpiration and photosynthetic rates. Horizontal airflow speeds above 1.0 ms^{-1} was suggested to achieve the maximal plants transpiration and photosynthetic rates of the crop canopy (Kitaya *et al*., 2003).

Physically measurement of airflow using sensors at different point around the crop is tedious and time consuming. It is also high in cost as it involves large quantity of sensors in order to provide detail airflow and temperature characteristics inside the plant factory, especially for a large scale plant factory. Therefore, the use of computer simulation is the solution. In this regard, computational fluid dynamics (CFD) is a computer simulation technique that has been shown to be an effective tool in simulating physical complex phenomena with reasonable accuracy. CFD has been widely used to study ventilation and climate uniformity in greenhouses (Bartzanas *et al*., 2004; Boulard & Wang, 2002; Bournet & Boulard, 2010; Lee *et al*., 2013; Tamimi *et al*., 2013). CFD studies to analyse ventilation in indoor plant factory are increasing (Baek *et al*., 2016; Lim & Kim, 2014; Moon *et al*., 2014) but further studies for evaluating air-distribution system design alternatives in indoor plant factory which are required to improve climate uniformity, especially for large-scale commercial indoor plant factories are still lacking. Therefore, this study was conducted with the focus on evaluating air-distribution system design alternatives tending to uniformity in respect to air temperature and airflow.

2. Materials and Methods

2.1. Data collection from existing plant factory for CFD validation

MARDI has successfully developed a plant factory with a size of 14.5 m width x 16.0 m length x 6.0 m height. The plant factory was occupied with nine unit of eight tier planting rack. The structure was totally enclosed. The climate inside the plant factory was controlled by an air-conditioner, fan, CO_2 inlet and outlet vent. The location of the inlet and outlet of the plant factory is as shown in Figure 1. Temperature and airflow data at the air conditioner inlet was measured to be the input value of the inlet properties in the CFD simulation process. In order to validate the CFD simulation results, temperature and airflow data at eight locations inside the plant factory as shown in Figure 2, was measured by using Sper Direct mini environment quality meter 850070.

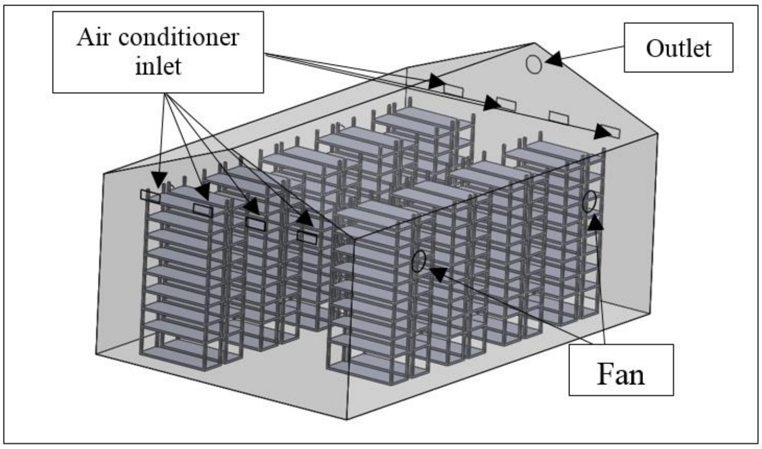


Figure 1. Inlet and outlet location.

Figure 2. Data collection location for validation.

2.2. CFD Simulation

A commercial CFD software Ansys Fluent was used for the simulations. The fluid flow in the plant factory domain was assumed to be a steady-state, incompressible with a three Dimension (3D) turbulent flow. The calculation of airflow was considered as mathematical formulations of the fluid mechanics conservation laws. By applying the mass, momentum, and energy conservation, the fundamental governing equations of fluid dynamics of the mass, momentum and energy equation can be written as Equation (1), (2) and (3), respectively:

Mass equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left[-\rho\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho C_a T) + \frac{\partial}{\partial x_j}(\rho u_j C_a T) - \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) = S_T \tag{3}$$

Where, $\boldsymbol{\rho}$ = fluid density (1.225 kgm-3) , t = time (s), x = Cartesian coordinates (m), i,j = Cartesian coordinates index, $\boldsymbol{u}$ = velocity component (m s^{-1}), $\boldsymbol{g}$ = acceleration due to gravity (-9.81 m s^{-2}), C_a = specific heat capacity (1006.43 Jkg^{-1} K^{-1}), T = temperature (K) and S_T = thermal sink or source (10000W m^{-3}). In this study, the realisable k-ε model was used to calculate the turbulent effect of air flow inside the plant factory. This turbulence model was chosen as many previous studies on CFD simulation for enclosed agriculture structure have shown that it was more accurate and possesses superior performance (Zhang *et al.*, 2016).

In this study, the 3D model of the plant factory was created using the solid works software. The 3D model was then imported to the ANSYS Fluent software for the simulation processes of the airflow and temperature distribution. The configurations for CFD simulation is as shown in Table 1.

For the current plant factory layout, the rack was located in front of the air conditioner inlet. In this arrangement, the rack has caused the blockage of cold airflow from the inlet which created uneven distribution of airflow inside the plant factory. Therefore, for the new design, the location of the air conditioner inlet was located in between the racks as shown in Figure 3. CFD simulation was carried out to analyse the airflow and temperature distribution for the new inlet configuration and compared with the existing configuration.

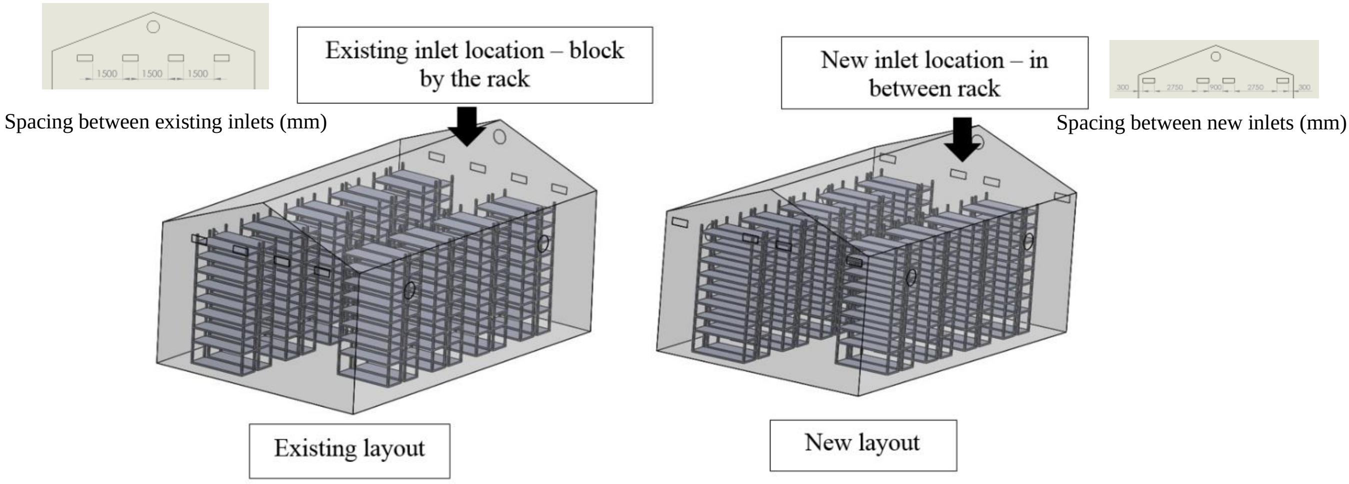


Figure 3. Comparison between existing and new inlet configuration of the plant factory.

Table 1. CFD simulation configurations.

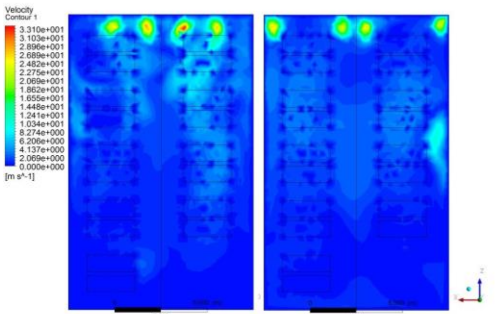
Parameter	Setting
Cell zone condition	
Wall	Solid – polyurethane and cement fibre board
Roof	Solid – polyurethane
Rack	Solid – material steel
Internal domain	Fluid – material air
Analysis type	Transient
Gravity	-9.81 ms^{-1}
Turbulence model	standard k-ε
Solar loading	
Longitude	101.97
Latitude	4.2
Time zone	+8
Solar direction	Data from solar calculator at 13:00 hours
Boundary condition	
Air cond inlet type	velocity-inlet
Air cond velocity	30.0 ms^{-1}
Air cond inlet temperature	18ºC
Fan inlet type	velocity-inlet
Fan velocity	10.0 ms^{-1}
Roof	Opaque, *no slip wall* Heat transfer: convention + radiation Solar radiation at negative –y direction, Radiation model- DO (*discreet ordinate*).
Wall	Opaque, *no slip wall* Heat transfer: convention + radiation Radiation model- DO (*discreet ordinate*).
Floor	no slip wall, fix temperature
Solution methods	SIMPLE (*semi-implicit pressure linked equation*)
Momentum	2nd order Upwind
Transient formulation	1st order implicit

3. Results and Discussions

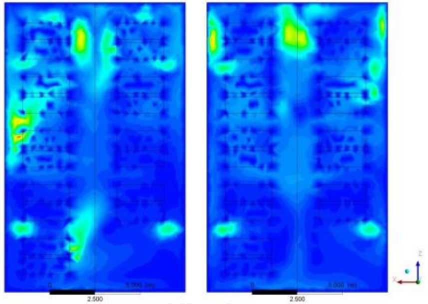
The validation process was carried out by calculating the root mean square error (RMSE) and the normalise root mean square error (NRMSE) between the measured data and simulated data at eight locations as shown in Figure 2. The RMSE and NRMSE between the measured and simulated data was 3.8% and 0.7%, respectively. As the NRMSE was below

10%, it can be concluded that the CFD simulation was able to represent the actual condition and can be used for further analysis.

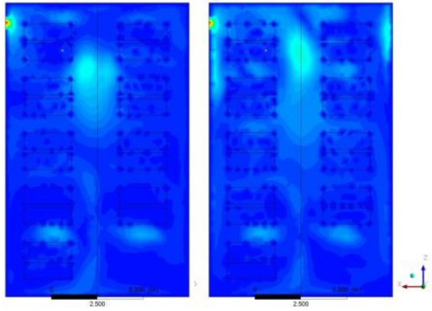
The airflow distribution plot for the existing inlet location and the new inlet location at the top, middle and bottom plane of the plant factory was as shown in Figure 4. Figure 5 shows the airflow distribution at the vertical cross-sectional plane i.e. at the left, centre and right plane inside the plant factory. Figure 4 and 5 show that the airflow was higher at the area close to the fan and air conditioner inlet for both the existing and current inlet locations. At the top and bottom plane, the airflow distribution was equal between the two configurations. However, at middle plane, the airflow distribution for the new inlet location was more evenly distributed compared to existing inlet location. In order to further compare the results between the existing and new inlet locations, airflow data from 15 locations (A1 to A3, B1 to B3, C1 to C3, D1 to D3 and E1 to E3) inside the plant factory was compared between the two configurations. The location of point 1 to 15 is as shown in Figure 6. Figure 7 shows all the airflow data for these 15 points. The data shows that the maximum difference between minimum and maximum airflow data for the new inlet location was lower at 3.1 ms^{-1} as compared to existing inlet location which was 4.2 ms^{-1}. The airflow uniformity inside the plant factory was evaluated with the coefficient of variation (CV) value. The CV is defined as the ratio of the standard deviation over mean. The CV shows the extent of variability in relation to the mean of the population. Therefore, a low value indicates a lower variability of the data set. For the airflow data at the 15 locations, the CV for the new inlet location was lower compared to existing inlet location which was 60% and 87%, respectively. These results showed that the airflow inside the plant factory with the new inlet location was more uniform compared to existing inlet location.



Top plane



Middle plane



Bottom plane

Figure 4. Comparison of airflow distribution in the plant factory between existing (left) and new (right) location of the inlet at different horizontal plane.

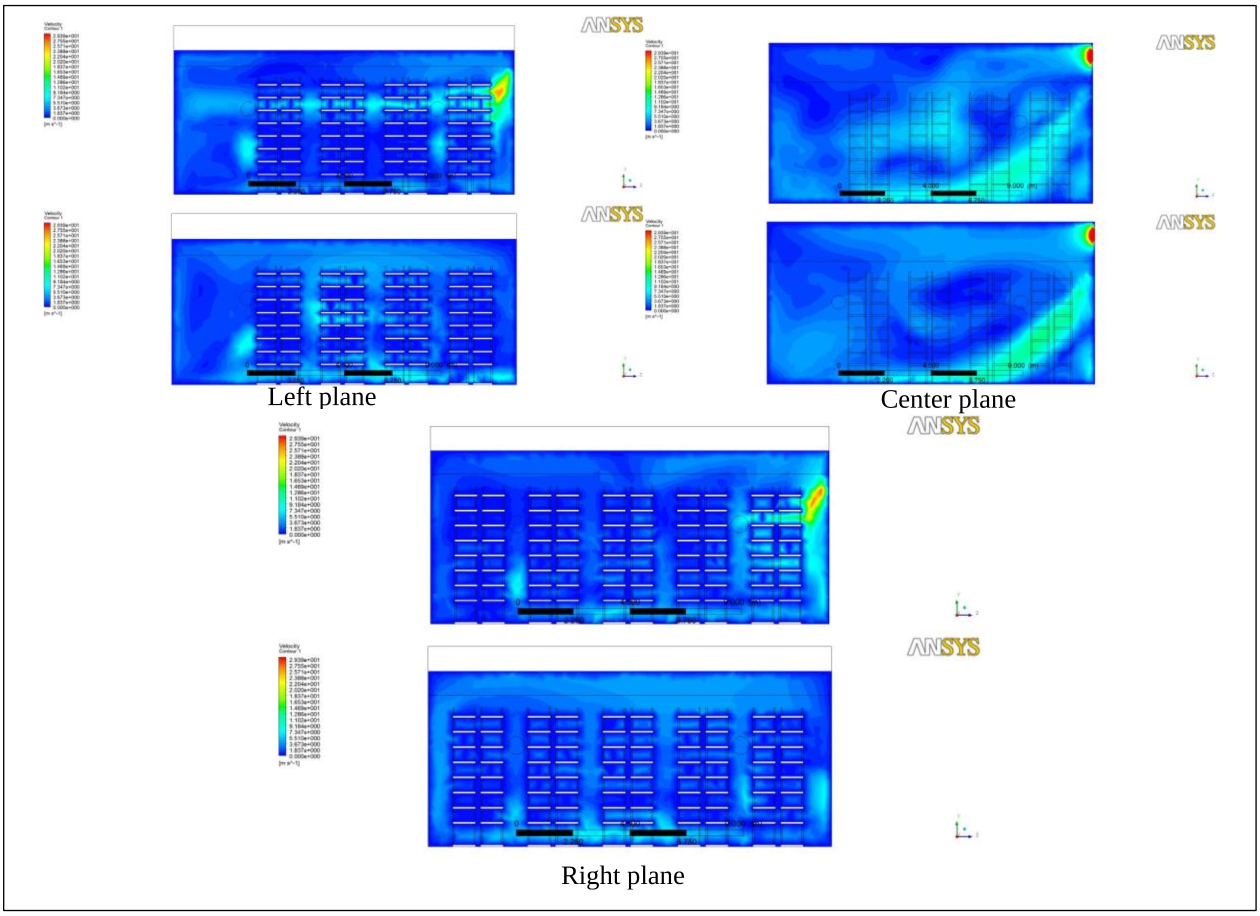


Figure 5. Comparison of airflow distribution in the plant factory between existing (top) and new (bottom) location of the inlet at different vertical plane.

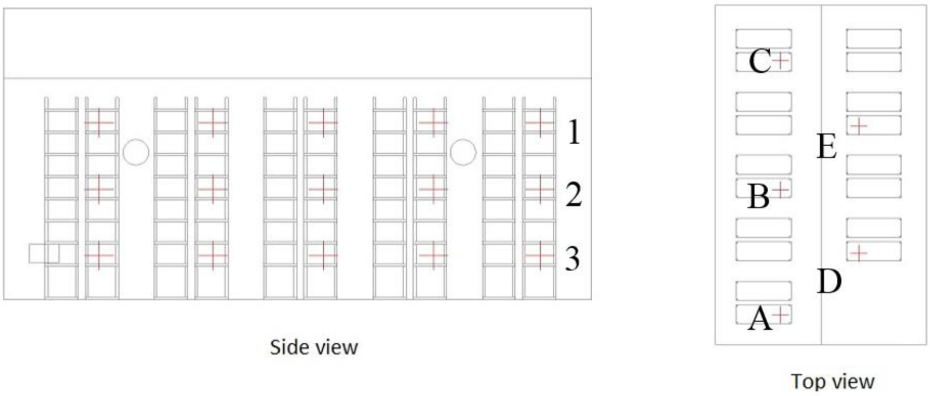


Figure 6. Location of data point A1 to E3 inside the plant factory.

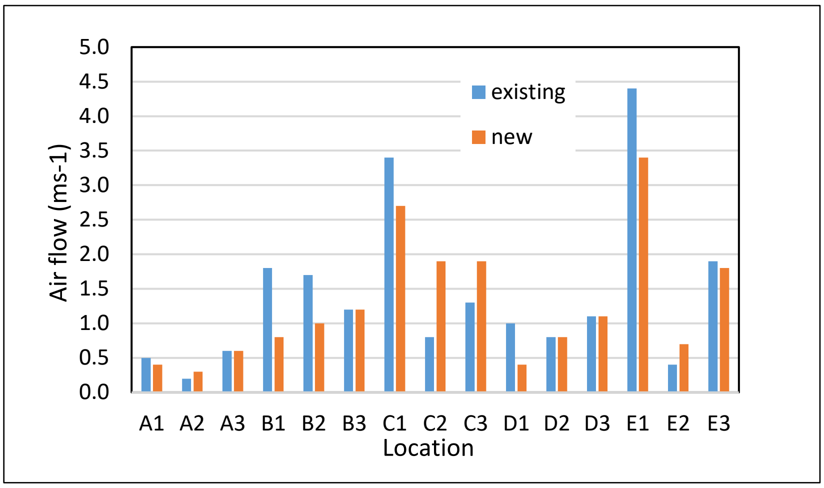
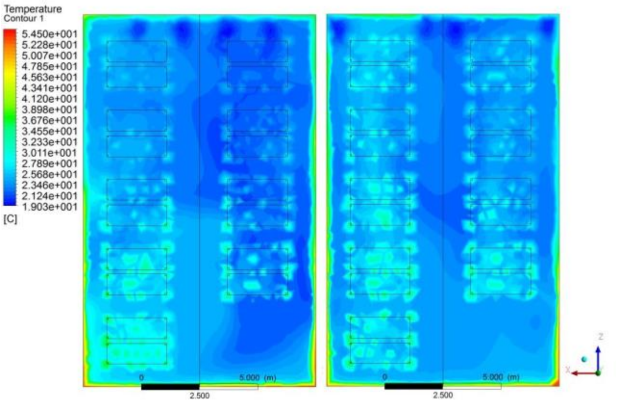
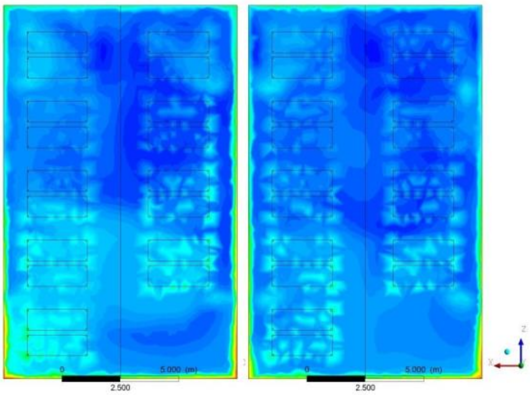


Figure 7. Airflow data at 15 locations inside plant factory.

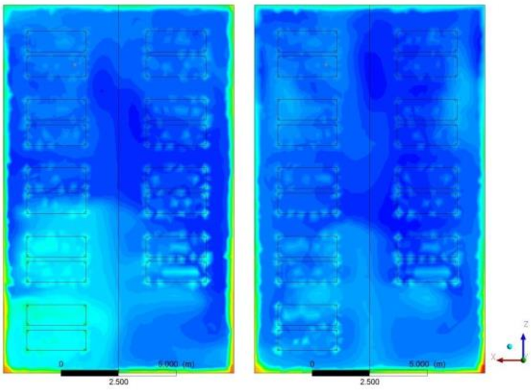
As the inlet is the cold air inlet, the location of the inlet also effect the overall temperature distribution inside the plant factory. The temperature distribution plot for the existing inlet location and new inlet location at the top, middle and bottom plane inside the plant factory is as shown in Figure 8. The figure showed that at the middle plane the temperature distribution between existing and new inlet locations were similar but at the top and bottom plane, the temperature distribution with the new inlet location was more uniform. Figure 9 shows the temperature data for the 15 locations inside the plane factory. The data showed that the maximum difference between minimum and maximum temperature for the new inlet location was lower at 5.1°C compared to existing inlet location which was 8.3°C. The mean temperature values of top, middle and bottom plane were 21.9, 21.8 and 20.6°C, respectively. Meanwhile, for the existing inlet location, the mean temperature value of top, middle and bottom plane were 22.9, 21.8 and 21.7°C, respectively. The temperature at the left area at the bottom plane for existing setup was higher compared to other area due to very low airflow at this area which was less than 0.6 ms^{-1} compared to other area which was more than 1.0 ms^{-1}. The CV for the temperature data of the new inlet location was lower compared to existing inlet location which was 7% and 18%, respectively. Therefore, the temperature distribution was found to be more uniform with the new inlet location as compared to existing inlet location.



Top plane



Middle plane



Bottom plane

Figure 8. Comparison of temperature distribution in the plant factory between existing (left) and new (right) location of the inlet at different level.

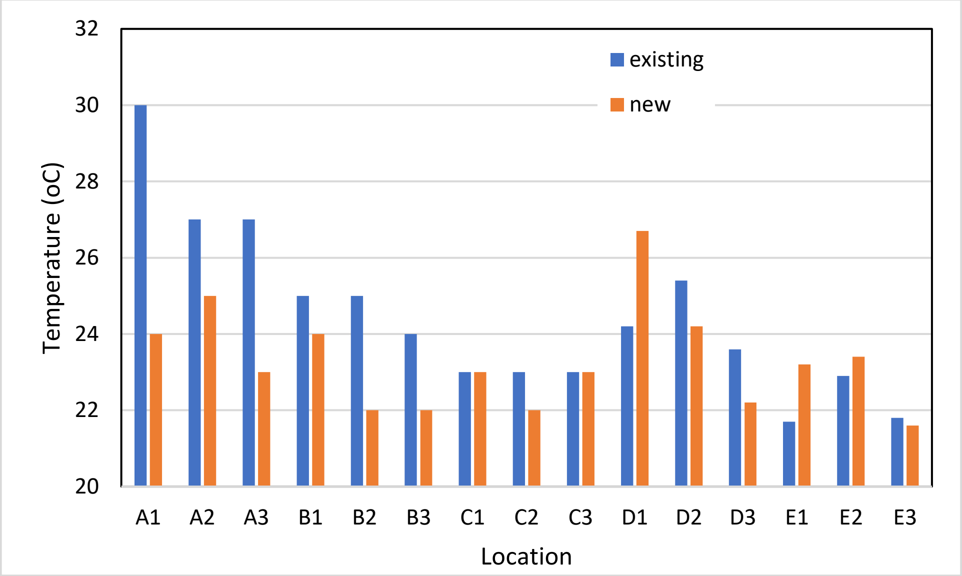


Figure 9. Temperature data for A1 until E3.

4. Conclusions

From the study, it can be concluded that the CFD simulation was able to represent actual condition with acceptable accuracy. Therefore, the CFD simulation can be used to analyse the effect of different design before the actual physical development. The study also concluded that changes in the inlet location affect the overall internal climate of plant factory. The results from the study proved that the new location of inlet was able to produce a more uniform airflow and temperature inside the plant factory compared to existing location.

References

Baek, M. S., Kwon, S. Y., & Lim, J. H. (2016). Improvement of the Crop Growth Rate in Plant Factory by Promoting Air Flow inside the Cultivation. *International Journal of Smart Home*, *10*(2), 63–74.

Bartzanas, T., Boulard, T., & Kittas, C. (2004). Effect of vent arrangement on windward ventilation of a tunnel greenhouse. *Biosystems Engineering*, *88*(4), 479–490.

Boulard, T., & Wang, S. (2002). Experimental and numerical studies on the heterogeneity of crop transpiration in a plastic tunnel. *Computers and Electronics in Agriculture*, *34*(1–3), 173–190.

Kitaya, Y., Tsuruyama, J., Shibuya, T., *et al*. (2003). Effects of air current speed on gas exchange in plant leaves and plant canopies. *Advances in Space Research*, *31*(1), 177–182.

Kozai, T., Niu, G., & Takagaki, M. (Eds.). (2015). *Plant factory: An indoor vertical farming system for efficient quality food production*. Academic Press.

Lee, I. B., Bitog, J. P. P., Hong, S. W., *et al*. (2013). The past, present and future of CFD for agro-environmental applications. *Computers and Electronics in Agriculture*, *93*, 168–183.

Lim, T. G., & Kim, Y. H. (2014). Analysis of airflow pattern in plant factory with different inlet and outlet locations using computational fluid dynamics. *Journal of Biosystems Engineering*, *39*(4), 310–317.

Moon, S. M., Kwon, S. Y., & Lim, J. H. (2014). Minimisation of temperature ranges between the top and bottom of an air flow controlling device through hybrid control in a plant factory. *The Scientific World Journal*, 2014.

Zhang, Y., Kacira, M., & An, L. (2016). A CFD study on improving air flow uniformity in indoor plant factory system. *Biosystems Engineering*, *147*, 193–205.