CFD SIMULATION IN HIGH-DENSITY PLANTING SYSTEM INSIDE A CONTAINER-TYPE PLANT FACTORY

Arina Mohd Noh
Engineering Research Center
Malaysian Agricultural Research and Development Institute (MARDI)
43400 Serdang, Selangor, Malaysia
Email: arina@mardi.gov.my

Fauzan Ahmad
Malaysia-Japan International Institute of Technology,
Universiti Teknologi Malaysia,
Kuala Lumpur, Malaysia
Email: fauzan.kl@utm.my

Mohd Zul Fadzli Marzuki
Engineering Research Center
Malaysian Agricultural Research and Development Institute (MARDI)
43400 Serdang, Selangor, Malaysia
Email: mfadzli@mardi.gov.my

Mohd Daniel Haziq Abdul Rashid
Engineering Research Center
Malaysian Agricultural Research and Development Institute (MARDI)
43400 Serdang, Selangor, Malaysia
Email: hazeq@mardi.gov.my

ABSTRACT

Computational fluid dynamics (CFD) plays an increasingly important role in designing the agriculture control environment structure in the past few years. A plant factory is a fully enclosed controlled environment agriculture structure developed to create optimum growing conditions for crops. Previous studies have proven that the CFD technique was able to analyse and predict the internal climate of the plant factory in the designing stage before the actual plant was built. Uniformity of airflow and temperature distribution is important in plant factories as it is responsible for creating an optimum and uniform growing condition for crops. The objective of the study is to find the best location of the air-conditioner and air outlet that produce the optimum air flow and cold air distribution inside the container size plant factory with a high-density planting. CFD simulation was conducted to analyse the changes in air flow characteristics and temperature distribution inside the plant factory with different locations of air-conditioner and air outlet before the actual plant factory was build. Three alternative locations i.e., location 1, 2 and 3 were simulated to improve the uniformity of airflow and temperature distribution. Temperature and airflow data was recorded and compare between these alternative locations. The location of the air-conditioner and air outlet that produced the optimum airflow and temperature distribution inside the plant factory was identified based on the alternative that produce lowest average temperature and higher air flow.

Keywords: *CFD simulation, plant factory, airflow distribution, temperature distribution*

INTRODUCTION

With dramatic population growth and resource scarcity expected in the coming decades, agriculture productivity is expected to only meet 10% of the food demand. Therefore, the development of new methods to increase the ratio of crop production to utilised land is of great importance. Moreover, agricultural lands are threatened not only by high urbanisation, uneven water distribution and adverse weather conditions but also by threats to biodiversity and adverse environmental impacts (Naranjani et al., 2022). Plant factory with high-density planting is the technique that shows a positive solution to this issue. High-density planting is an agricultural practice that involves planting crops at a higher density than is traditionally used. The goal of high-density planting is to maximise the use of available space and resources, increase crop yields, and improve overall efficiency. In high-density planting, crops are planted in a closely spaced pattern, often with multiple plants per hole or row. This allows for more plants to be grown per unit area of land, resulting in a higher yield. It can also help to reduce soil erosion, weed growth, and the need for chemical inputs such as fertilizers and pesticides. While high-density planting can provide many benefits, it also requires careful management of resources to ensure that the plants have enough resources to grow and produce. In the enclosed structure of crop production such as a plant factory, high-density planting will affect the distribution of airflow and temperature inside the structure.

Understanding the internal airflow in a plant factory in detail is important to effectively deliver cold air to the planting area to maintain climatic uniformity and promote adequate air movement around the crops. In high-density planting, the uniformity of the temperature and airflow inside the plant factory can be lowered due to the increased number of layers of cultivation shelves, the increased resistance to airflow by plants, and the excessive heat generated from artificial lighting sources (Lim and Kim, 2014).

Therefore, before designing the plant factory, the analysis of the airflow characteristics is a very important task to improve the uniformity of air temperature and humidity distribution. A study by Kozai and Takagaki (2015) proved that air movement plays an important role in aerodynamics at the leaf surfaces. It affects the gas, heat, and water exchange of plants and thus affects plant transpiration and photosynthetic rates

Locations of the cold air inlet and outlet and airflow speed from the inlet are very important factors for the environmental control of the closed plant production system. Particularly, the overall airflow characteristics can be greatly affected by the locations of the inlet and the outlet. A study by Fang et al., (2020) indicates that to enhance the airflow inside the crop interior, it is necessary to conduct further research by using different locations and airflow angles of the air duct. Previous studies by Zhang et al., (2016) and Niam et al., (2019) on the airflow pattern of a closed space in the shape of a box, with different locations and sizes of the inlet and the outlet, have reported the obvious difference in airflow characteristics.

Physically measuring the temperature and airflow using sensors at different points around the crop is tedious and time-consuming. It also involved a high cost. Therefore, computer simulation is the solution. Computational fluid dynamics (CFD) is a computer simulation technique that is an effective tool for simulating complex physical phenomena with reasonable accuracy. CFD is a branch of fluid mechanics that deals with the numerical analysis of fluid flow and heat transfer (Stanbury et al., 2017). It involves the use of computer algorithms and numerical methods to solve complex equations that describe the behaviour of fluids. In CFD, the fluid is discretised into a large number of small cells or elements, and the equations governing the flow of fluid are applied to each of these cells. The equations can include the conservation of mass, momentum, and energy, as well as various constitutive relationships that describe how the fluid behaves. CFD simulations can be used to study a wide range of fluid flow problems, such as the behaviour of fluids in pipes, the flow of air over an aircraft wing, or the mixing of fluids in a chemical reactor. They can also be used to predict the behaviour of fluids in real-world situations, such as the flow of air around buildings, the spread of pollutants in the atmosphere, or the flow of blood through arteries. The results of CFD simulations can provide detailed information on the behaviour of fluids that may be difficult or impossible to obtain experimentally. They can also be used to optimise the design of engineering systems, improve the efficiency of processes, and reduce the cost of development. In plant factory design, a study by Suwardana et al., (2022) concluded that the CFD simulation is feasible for analysing the airflow distribution in the mini plant factory with an error value of 4.39%. Another study by Zhou et al, (2020) concluded that the measured temperature values of the plant factory had a good agreement with the simulated values by the CFD model with an RMSE value of 3%.

Many studies have used CFD simulation to analyse ventilation and temperature in the indoor planting system (Naranjani et al., 2022; Natarajan et al., 2022, Noh et al., 2021; Mohd Noh, et al., 2020; Fan et al., 2020; Niam et al., 2019) but further studies for evaluating air-distribution system design alternatives in the indoor plant factory are required to improve climatic uniformity, especially for high-density planting. Therefore, this study is focused on evaluating cold air-distribution system design alternatives inside the shipping container-size plant factory concerning air temperature and airflow, especially for high-density planting systems. The main objective of the study is to find the best location for air-conditioner and outlet air that produces the lowest average temperature and highest average airflow inside the plant factory. The save time and cost, this process needs to be carried out before the actual plant factory is developed. In this study CFD simulation tool was used the analyse the temperature and airflow distribution changes virtually with different combinations of air-conditioner and air outlet locations.

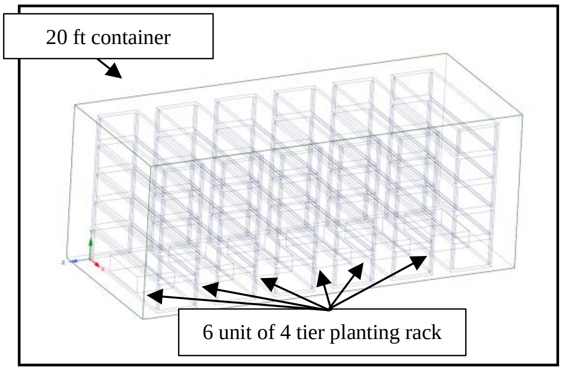
MATERIALS AND METHODS

CAD Model of High-Density Planting System

Malaysia is a tropical country with maximum daily temperatures hovering between 32.2 - 35°C. This condition is not suitable for lettuce growing. Therefore, a plant factory with a control environment system was developed to enable the planting of lettuce in Malaysia’s low land area. Modifying shipping containers to become plant factories is one of the options to build a plant factory with a minimal capital cost. In addition to this, to increase the quantity of lettuce planted per unit area and maximize the yield, a high-density planting system was selected and to further reduce the production cost of each plant.

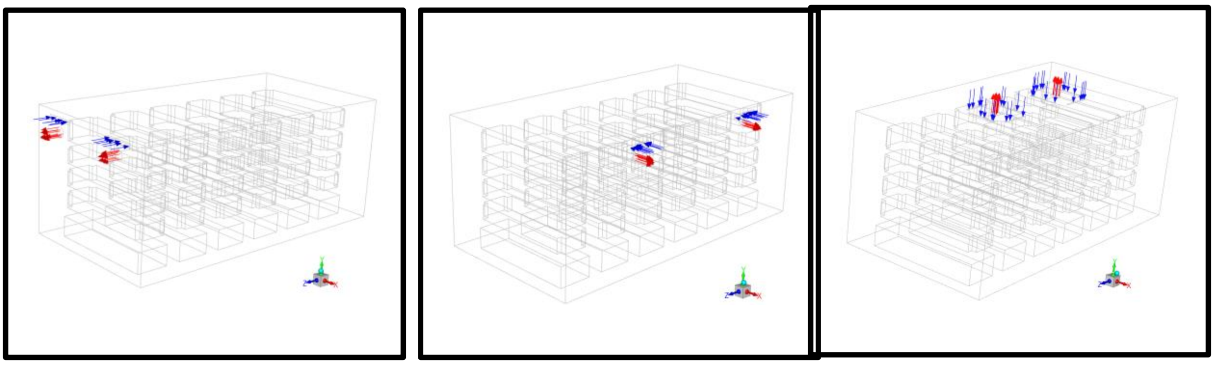
A 20 feet container size was used in the design. The plant factory was equipped with 6 units of 4-tier planting racks. Figure 1 shows the layout of the high-density plant factory. The size of each planting rack is 2.4 m height x 0.6 m width x 2.3 m length. The estimated production capacity of the system is 6000 plants per cycle.

Figure 1, layout of the high-density plant factory



Two air-conditioning units with two exhaust fans were designed to be placed inside the plant factory to produce the optimal temperature for lettuce growing. A previous study has shown that the optimum temperature range for lettuce to grow is between 10.8 ºC and 29.7 ºC (Gent, 2017). In this study three locations of the air conditioner (cold air inlet) and the exhaust fans (outlet) were selected. These three locations were selected referring to the place of air-conditioner normally installed for a room. The air conditioner type used was the cassette type for locations 1 and 2 and the ceiling type for location 3. The 3-dimensional (3-D) CAD model of the design was developed using commercial CAD software Ansys SpaceClaim. The 3-D model is shown in Figure 2. For location 1 the cold air inlet was located at the short side wall and the outlet was located below the inlet. For location 2 the cold air inlet was located at the long side wall and the outlet was located below the inlet as well finally for the third location the cold air inlet was placed on top of the container while the outlet location was located below the inlet.

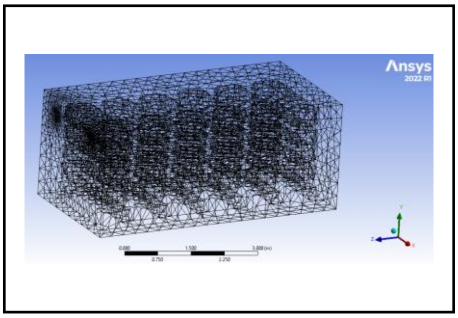
Figure 2, location of air conditioner (blue arrow) and exhaust fan (red arrow)



(a) Location 1 (b) location 2 (c) location 3

The CAD model of the system was then imported into the Ansys meshing software. A tetrahedral type of mesh was used in the study and the total number of elements was 26937 with a mesh size of 0.3 m. Figure 3 shows a sample of mesh used in the study for location 1.

Figure 3, an example of mesh for location 1



CFD Simulation

In this study, the fluid flow in the plant factory was assumed to be a steady-state, incompressible, and three-dimensional turbulent flow. The numerical calculation of the temperature and airflow is generally based on the conservation equations of aerodynamics on (3) mass, (4) momentum, and (5) energy using the Navier-Stokes equations. The Navier-Stokes equations are highly nonlinear and generally cannot be solved analytically. Instead, they are typically solved numerically using computational fluid dynamics (CFD) techniques. Despite the challenges in solving them, the Navier-Stokes equations are widely used in engineering, physics, and other fields to model fluid flows in a variety of applications. In CFD simulation, enforcing these conservation laws over discrete spatial volumes in a fluid domain, it is possible to achieve a systematic account of the changes in mass, momentum and energy as the flow crosses the volume boundaries. In Ansys Fluent the resulting equations can be written as below (Fluent, 2011):

Continuity equation: $\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = S_M$ (3)

Momentum equation: $\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\overline{\overline{\tau}}) + \rho\vec{g} + \vec{F}$ (4)

Energy equation: $\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = \nabla.\left[k_{eff}\nabla T - \sum_j h_j\vec{J}_j + (\overline{\overline{\tau}}_{eff}.\overrightarrow{v})\right] + S_h$ (5)

The first three terms on the right-hand side of Equation (5) represent an energy transfer due to conduction, species diffusion, and viscous dissipation, respectively.

The turbulent kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] G_k + G_b - \rho\varepsilon - \gamma_M - S_k \tag{6}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{7}$$

The 3D mesh model of the design was imported to the ANSYS fluent software for the simulation process of the airflow and temperature distribution. The configurations for CFD simulation are shown in Table 1.

Table 1. CFD simulation configurations

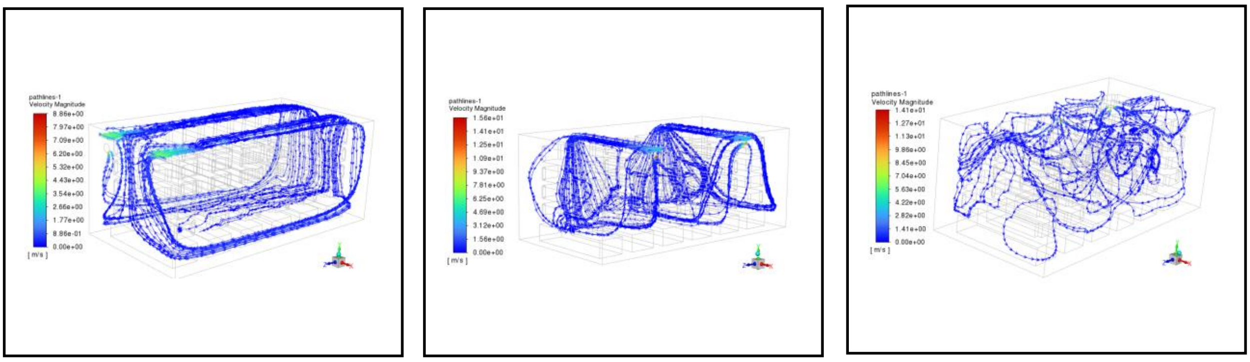
Parameter	Setting
Cell zone condition	
Internal domain	Fluid – incompressible air
Analysis type	Steady-state
Gravity	-9.81 ms^{-1}
Turbulence model	standard k-ε
Boundary condition	
Air conditioner inlet type	velocity-inlet
Air conditioner velocity	5.0 ms^{-1}
Air cond inlet temperature	16^{o}C
LED heat	100 Wm^{-2}
Floor	no slip wall, fix temperature
Wall	No slip wall, thermal mix (convection and radiation)
Solution methods	SIMPLE (*semi-implicit pressure linked equation*)
Momentum	2nd order Upwind

RESULT AND DISCUSSION

The airflow distribution for all options of inlet and outlet location is shown in Figure 4. Figure 4, shows that when the air condition was placed on the side wall as in location 1 and location 2, the cold air from the air-conditioning flowed directly to the wall at the other end and the air flowed downward along the adjacent wall before flowing back in between the growing rack layer. Due to the loss of airflow velocity as it travels further from the cold air source, the airflow was not uniformly distributed in between the growing rack layers, especially in the middle of the growing racks. The flow is higher at the bottom layer compared to the middle and upper layers. For the location 3 option, the airflow was more uniformly distributed from all sides. This is because each ceiling-type air conditioner had 4 sides of cold air outlets, therefore the cold air was much more evenly distributed between the growing racks. However, due to the narrow space between racks, the airflow was blocked by the rack itself.

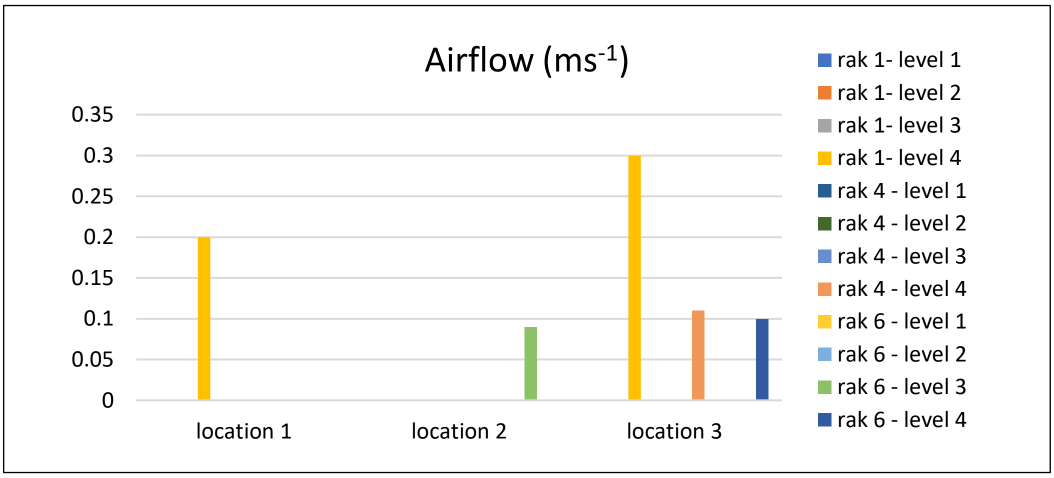
To further compare the results between all locations, air flow data at each layer of growing rack numbers 1, 4 and 6 were recorded and compared. Figure 5 shows the airflow data for these 12 locations. The data shows that for location 1 and location 2, many points did not record any airflow. Only at the bottom layer, the airflow was recorded which is 0.2 ms^{-1} for location 1 and 0.09 ms^{-1} for location 2 respectively. This is due to the uneven flow of cold air in the area of the growing rack as mentioned above. While for location 3 more locations recorded air flow with values ranging from 0.1 to 0.3 ms^{-1}.

Figure 4, air flow velocity plot



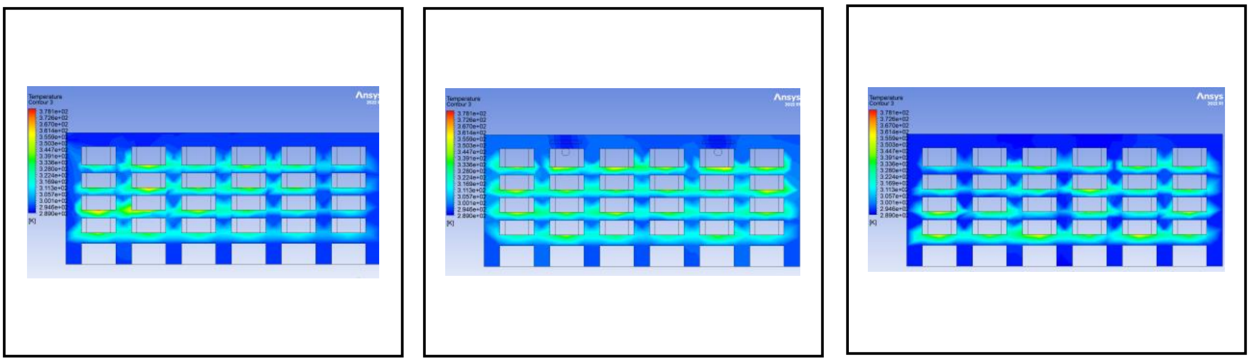
(a) location 1 (b) location 2 (c) location 3

Figure 5, air flow data in between layer



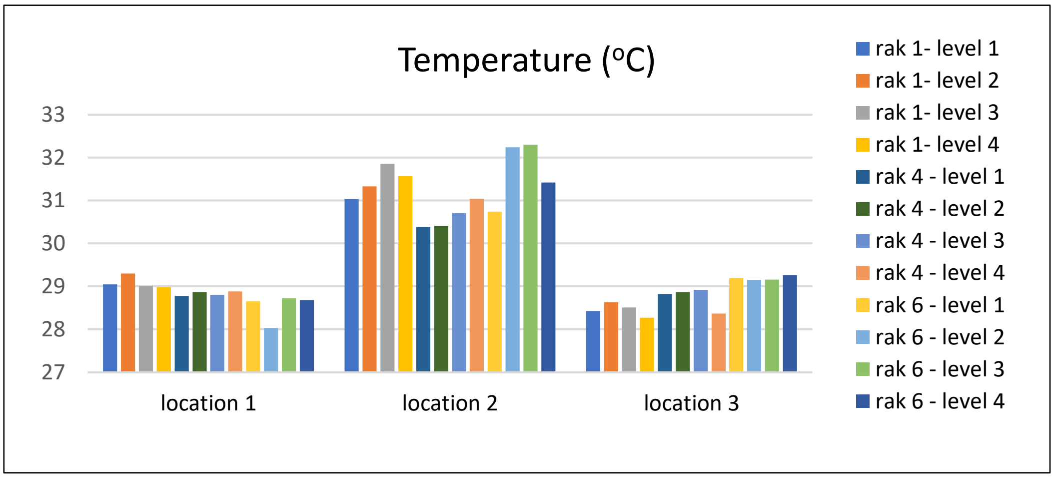
The temperature distribution at the centre plane of the plant factory for all locations is shown in Figure 6. The temperature distribution plot was similar for all locations where the temperature in between the layers of the growing racks was higher due to the heat produced by the LED. To further compare the results between all locations, temperature data in between layers of growing rack numbers 1, 4 and 6 were recorded and compared. Figure 7 shows the temperature data for these 12 locations. The data shows that for location 1 the temperature was between 28.0 and 29.3 °C. For location 2 the temperature was between 30.3 and 32.3 °C while for location 3 the temperature was between 28.3 and 29.2 °C. The average temperature for locations 1, 2 and 3 were 28.8, 31.3 and 28.7 °C respectively. These data show a relation between the temperature and the airflow data. It proved that locations 1 and 3 where the airflow is higher in more data points it able to produce lower average temperatures. For location 2, although it is similar to location 1 where only one point recorded airflow, the average temperature is higher as the cold air flowing at the bottom area is lower which is only 0.08 ms^{-1} compared to 0.2 ms^{-1} for location 1.

Figure 6, temperature distribution contour plot



(a) location 1 (b) location 2 (c) location 3

Figure 7, temperature data in between layer



The result shows that CFD simulation was able to simulate the changes in the airflow and temperature distribution inside the plant factory with different cold air inlet and outlet locations. By using CFD simulation, the optimum airflow and temperature distribution can be determined before the actual plant factory is built, this reduces the cost and time if the process was carried out after the actual plant factory was developed. In this study, the detail of the airflow and temperature data can easily be determined although it is inside the high-density planting with many planting racks.

CONCLUSION

In this study, the CFD simulation was used to investigate the impact of different air conditions and exhaust fan locations on the airflow and temperature characteristics inside a high-density container-size plant factory. The study concludes that the airflow and temperature inside the plant factory are affected by changes in the air condition and exhaust fan locations. The study's findings also revealed that the air conditioner and exhaust fan located in locations 1 and 3 were able to produce the temperature distribution within the range which is optimum for lettuce growing. From the study, it was also proven that with proper design of the cold air distribution system i.e., the correct placement of air conditioner and exhaust fan, a container-size plant factory with a high-density planting method was able to produce optimum conditions for lettuce growing.

REFERENCES

Fan, R., Liu, H., Zhou, S., He, Z., Zhang, X., Liu, K., ... & Lu, W. (2020, August). CFD simulation of the airflow uniformity in the plant factory. In *IOP Conference Series: Earth and Environmental Science* (Vol. 560, No. 1, p. 012074). IOP Publishing.

Fang, H., Li, K., Wu, G., Cheng, R., Zhang, Y., & Yang, Q. (2020). A CFD analysis on improving lettuce canopy airflow distribution in a plant factory considering the crop resistance and LEDs heat dissipation. *Biosystems Engineering*, *200*, 1-12.

Fluent, A. N. S. Y. S. (2011). Ansys fluent theory guide. ANSYS Inc., USA, 15317, 724-746.

Gent, M. P. (2017). Factors affecting the relative growth rate of lettuce and spinach in hydroponics in a greenhouse. *HortScience*, 52(12), 1742-1747.

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

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

Mohd Noh, A., Mohamad Tahir, M.A., Mat, S. and Dzulkifli, M.H. (2020). CFD simulation of temperature and airflow inside a shipping container-size plant factory for optimal lettuce production. *Food Research 4 (Suppl. 6): 54 – 59*

Naranjani, B., Najafianashrafi, Z., Pascual, C., Agulto, I., & Chuang, P. Y. A. (2022). Computational analysis of the environment in an indoor vertical farming system. *International Journal of Heat and Mass Transfer*, *186*, 122460.

Natarajan, G., Zaid, M., Konka, H., Srinivasan, R., Ramanathan, S. S., Ahmed, T., & Chowdhury, H. (2022, November). Modeling of air distribution inside a shipping container plant factory using computational fluid dynamics (CFD). In *AIP Conference Proceedings* (Vol. 2681, No. 1). AIP Publishing.

Niam, A. G., Muharam, T. R., Widodo, S., Solahudin, M., & Sucahyo, L. (2019). CFD simulation approach in determining air conditioners' position in the mini plant factory for shallot seed production. In *AIP Conference Proceedings* (Vol. 2062, No. 1, p. 020017). AIP Publishing LLC.

Noh, A. M., Noor, H. M., & Ahmad, F. (2021). CFD Simulation of the Airflow Distribution Inside Cube-Grow. *CFD Letters*, *13*(12), 81-89.

Stanbury, P. F., Whitaker, A., & Hall, S. J. (2017). Design of a fermenter. *Principles of Fermentation Technology*, 401–485.

Suwardana, R., Nugroho, A. P., Prasetyatama, Y. D., Falah, M. A. F., Sutiarso, L., & Okayasu, T. (2022, December). Analysis of airflow distribution on urban mini plant factory using computational fluid dynamics. In *IOP Conference Series: Earth and Environmental Science* (Vol. 1116, No. 1, p. 012029). IOP Publishing.

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.

Zhou, S., Zhang, X., He, Z., Liu, H., Liu, K., Chen, Y., ... & Lu, W. (2020, August). Experimental validation of CFD model with the air velocity and temperature in the plant factory. In *IOP Conference Series: Earth and Environmental Science* (Vol. 560, No. 1, p. 012089). IOP Publishing.