Using CFD to improve flow conditions in vertical farms using realistic plant geometries.

Ir. W. Plas[1,2], Prof. dr. Ir. M. De Paepe[1,2]

[1]Department of Electromechanical, Systems and Metal Engineering, Ghent University, Campus UFO, T4, Sint-Pietersnieuwstraat 41, 9000 Ghent, Belgium

[2]FlandersMake@UGent corelab EEDT MP Flanders Make, Gaston Geenslaan 8, 3001 Heverlee, Leuven, Belgium

Abstract

Ensuring optimal climate conditions in vertical farms is essential, as these farms generally operate in more confined conditions than in classical greenhouses. Computational Fluid Dynamics (CFD) has been used in the past to visualize flow fields in greenhouse type geometries. These CFD models represent the plant as porous blocks, wherein momentum, water vapour and energy source terms are added to mimic the behaviour of the plant. Vertical farms operate in a much more confined environment, and representing the plants as porous blocks can simplify the flow field too much. Therefore, in this study a realistic plant geometry is used in a realistic vertical farming environment to study the flow over these plants. A basil like plant structure is used to model the plant. Two systems are compared, a sideway ventilated vertical farm with nozzles, and a top ventilated vertical farming system with nozzles. Plant transpiration and energy exchange is implemented by setting the appropriate water vapour and temperature boundary conditions on the sides of the leaves. By using a realistic plant geometry, water vapour fluxes and energy fluxes can be directly calculated for every leaf. Both cases are compared by looking at the water vapour and heat transfer coefficients. Additionally, the average velocities around the leaves of the plant can be calculated as well, and these show that the top ventilated plants have a higher percentage of velocity within a certain range compared to the sideway ventilated configuration.

Keywords: Basil, CFD, Vertical Farming, Ventilation system

Introduction

In order to have a more sustainable food production in the future and to have the ability to grow food in urban environemnts, researchers are looking at cultivating crops in vertical farms or plant factories. A plant factory is a closed off environment in which artificial lighting is used in the form of LED lighting and the environment around the plant is accurately controlled. This has the benefit that more water can be recuperated, less CO2-enrichment needs to be used and that crops are grown closer to the consumers in urban environments [1].

As these crops are grown in a closed environment, a good ventilation is necessary in order to avoid adverse growing conditions at the crops. Furthermore, several studies have shown that an increased air flow rate corresponds with an increased transpiration rate and photosynthetic rate [2].

Computational Fluid Dynamics (CFD) has proven to be a useful tool to visualise flow fields inside a plant factory. Zhang et al. (2016) for example studied air flow inside a plant factory, which is ventilated from the top using perforated tubes [3]. In the paper of Zhang et al. (2016), different configurations of tubes and jets were investigated and were compared by looking at what percentages of flow are within a given range of velocities. Plants were modelled as a roughness height at the bottom floor and plant architecture was completely omitted. Fang et al. (2020) modelled air flow inside a lettuce plant factory as well [4]. Air flow was delivered to the plant factory using jets positioned at the side of the domain. Different configurations of diameters and hole spacings were investigated. Plants were modelled as a porous zone in the paper of Fang et al. (2020). This means that an extra empirical sink-term in the momentum-equation is added, which slows down the flow. The effects of lettuce transpiration and LED-light heating are included by adding an extra source-term in the energy equation. This approach however needs a priori knowledge of transpiration and LED-lighting data. Transpiration of the crops needs to be known before the simulation can be started. Furthermore, in the paper of Fang et al. 2020, no temperature plots or temperature simulation data is shown. Recently, Naranjani et al. (2022) has investigated the environment in an indoor vertical farming system using CFD [5]. The used crop in this study is lettuce. Plants are modelled as porous zones, and water vapour transport, CO_2, and O_2 transport are all modelled by adding an extra fixed source term to the respective transport equation. Different ventilation configurations are tested and are compared by looking at the uniformity of the flow in the zone around the plants. Different configurations of inlets and outlets are compared. Plants are once again modelled as porous zones, and specific plant architecture is thus omitted. This has the benefit that meshes do not get too complicated but has the disadvantage that air flow around the plants is not accurately captured. Furthermore, experimental values, such as drag coefficients or transpiration values need to be implemented in these porous zones to accurately model plant behaviour. In our study, a realistic plant geometry is used to study different ventilation configurations inside a realistic plant factory geometry.

Materials and Methods

The aim of our study is to investigate two different ventilation methods using CFD by looking at several quality parameters, such as the uniformity of the flow in the zone around the plants and the distribution of the individual convection coefficients and aerodynamic resistances around the leaves. Two different mass flow rates are simulated per case, giving a total of four studied cases.

Plant geometry creation

The plant used in this study is based on a basil plant, as this plant has a relatively easy to describe geometry. Experimental growing data of this plant is also available and collected in a previous experiment. The plant is made up of several leaves grouped together and ordered centrally along a

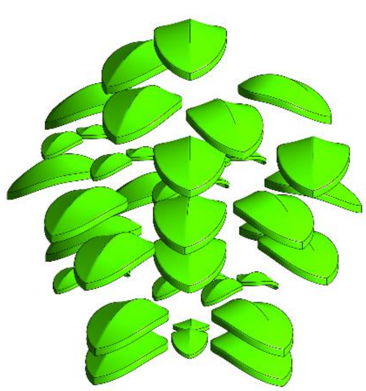

Figure 1a: Plant geometry used in the CFD simulations

Figure 1b: Actual plant model on which the CFD model is based

vertical axis. In order to save computational cost, only half a plant is modelled. In Figure 1a the plant in the CFD model is shown and in Figure 1b, the basil plant on which the virtual plant is modelled is shown.

The leaf itself was modelled using an analytical expression, as proposed by Coussement et al. 2018 [6]. In Table 1, the overall dimensions of the measured basil plant and the CFD model plant are listed. Care was taken that the total leaf area resembles the measured leaf area. The total number of leaves in the CFD model is smaller than the measured number of leaves. This can be attributed to the fact that the measured basil plant has a lot of young small leaves, which are neglected in the CFD model. In the CFD model, less leaves with a bigger leaf area are thus used to represent the plant model.

Table 1: CFD plant model and measured basil plant geometric parameters

	CFD plant model (1/2 plant)	**Measured (full plant)**
Number of leaves [-]	39	217$\pm$39
Leaf area [m^2]	0.0259	0.0520$\pm$0.0048
LAI [$m^2 m^{-2}$]	2.723	3.67$\pm$0.55
LAD [$m^2 m^{-3}$]	20.91	21.88$\pm$4.16
Plant height [m]	0.1305	0.17$\pm$0.01

Simulation Domain

The simulation domain consists of eight plants ordered vertically side by side in a rectangular box. The rectangular box has a length of 1380mm, a height of 350mm and a width of 80mm. The plants are placed 150mm apart from one another. Two different ventilation strategies are investigated, flow from the side using perforated jets, Case 1 and flow from the top using perforated jets, Case 2. In Case 1, air is delivered to the domain using two holes with a diameter of 22.5mm, situated 100mm from each other. In Case 2, air flow is delivered to the domain using ten jets placed at the top with a diameter of 10mm with a spacing of 120mm. The inflow velocity in both cases is chosen so that it resembles the air flow rate, used in the paper by Fang et al. 2020. Cases 3 and 4 are analogous to Cases 1 and 2 in the fact that they have twice the flowrate as Case 1 and 2 respectively. The inlet temperature for all cases is 303.15K or 30°C and the inlet mass fraction of water vapour is set at 0.0172kg water vapour/kg air. An overview of the four simulated cases and the case, used by Fang et al. 2020 is shown in Table 2.

Air flow leaves the domain through several pressure-outlet. Since these two ventilation methods are inherently different, the place of the pressure-outlets is different as well. For Case 1 and 3, air leaves the domain at the back of the domain, as shown in Figure 2 for Case 2 and 4, air is able to leave the domain in front and at the bottom of the domain.

There is a symmetry plane in the plane, where the plants are cut in half and a symmetry plane at the other side as well. All other walls are no-slip walls, representing the domain of a realistic plant factory.

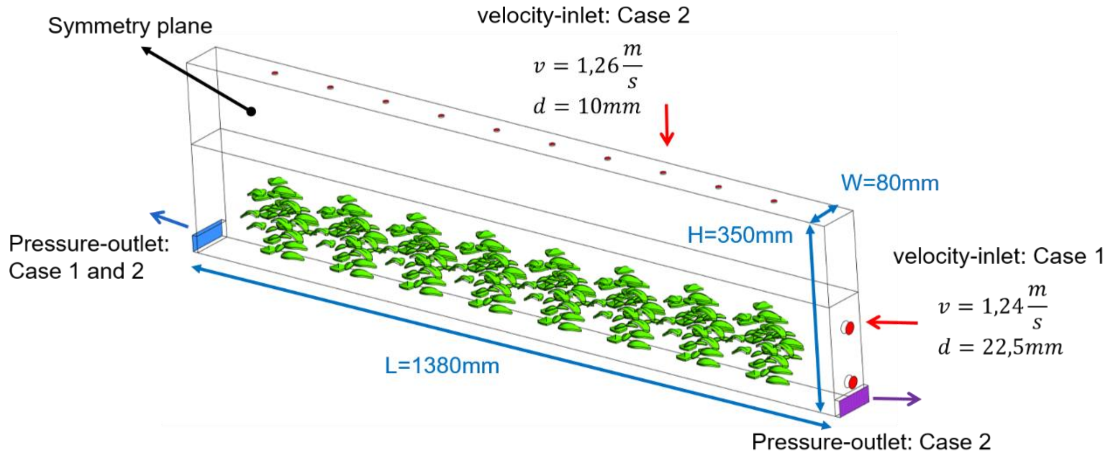

Figure 2: Simulation domain

Table 2: Overview of simulated cases

	Ventilation type	**Velocity-inlet value [in m/s]**	**Volume flow rate [in m^3/h]**	**Hole diameter (in mm)**	**Number of holes**
Case 1	Jets from the side	1.24	2.218	22.5	2
Case 2	Jets from the top	1.26	2.218	10	10
Case 3	Jets from the side	2.48	4.436	22.5	2
Case 4	Jets from the top	2.52	4.436	10	10
Fang et al. 2020		5.12	2.218		

Plant model

Plants cool down by transpiring water and heat up by receiving radiation from either the sun, or an artificial light source, such as LED-lamps. This behaviour could be implemented by using a variable heat flux boundary condition at the upper side of the leaf. We are however interested in choosing the best configuration for plant ventilation and it is therefore chosen to put both sides of the leaves at a constant temperature of 299.65K or 26.5°C. Using this method, it is possible to calculate in CFD the heat that is transferred from each leaf to the environment. An extended model where this temperature is a function of the simulated transpiration and incoming radiation can be a future adaptation to the model.

Transpiration is modelled by putting the underside of the leaf at a certain fixed absolute humidity, as most of these stomata can be found on this underside of the leaf. A specialized User Defined Function is written which is able to calculate the water vapour mass transfer, or transpiration of each individual leaf. The transpiration of the leaf is now governed by the value of the water vapour mass fraction at the underside of the leaf. The water vapour boundary layer around the leaves is governed by the aerodynamic resistance or its inverse the aerodynamic conductance. In Equation 1, the transpiration of each individual leaf is written, where Tr is the transpiration rate (in kg/s), A_{leaf} is the individual leaf area (in m²), ρ is the density of air (in kg/m³), ω_{leaf} is the humidity at the underside of the leaf and ω_{in} is the humidity at the inlet of the domain.

Equation 1: Transpiration from a single leaf

$$\frac{Tr}{A_{leaf}} = \rho \frac{\omega_{leaf} - \omega_{in}}{r_a}$$

This equation can now be rewritten in function of the unknown aerodynamic resistance, r_a, as shown in Equation 2. This aerodynamic resistance is the inverse of the aerodynamic conductance and is a measure of how the mass boundary layer at the underside of the leaves behaves. A high value indicates that a large mass boundary layer is present and thus also a bad water vapour transfer. Low values indicate the opposite, namely smaller mass boundary layers and good water vapour transfer.

Equation 2: formula for calculating aerodynamic resistance

$$r_a = \rho \frac{A_{leaf}}{Tr} \left(\omega_{leaf} - \omega_{in}\right)$$

Results and discussion

The aim of this study is to use an actual plant geometry to study different ventilation principles inside a plant factory and to compare these to each other. Four different cases are listed and are compared to one another by looking at the uniformity of flow around the plants and by looking at the distribution of aerodynamic resistances of the individual leaves.

Velocity, temperature and water vapour contours

Velocity, temperature and water vapour contours are shown in Figures 4a and 4b for Case 1 and 2 in a plane through the jets. The position of the plane through the jets is shown in Figure 3. Cases 3 and 4 have twice the flowrate as Cases 1 and 2 and are not shown, as these have similar distributions of velocity, water vapour and temperature. Temperature and water vapour are more saturated at the last plants for Case 1, where the plants are ventilated sideways. Local velocities are lower there, and water vapour and temperature are thus more accumulated over there.

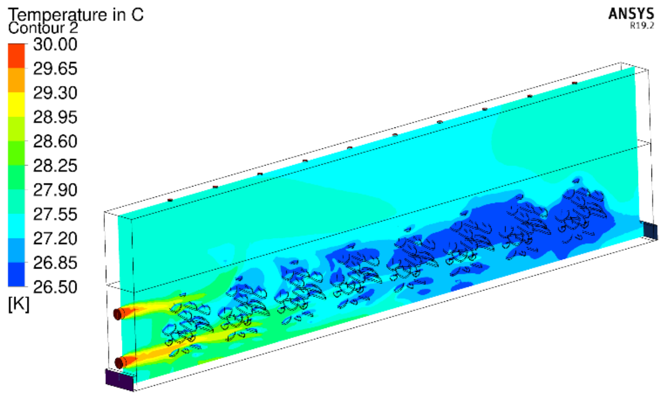

Figure 3: Position of the plane through the jets in the simulation domain

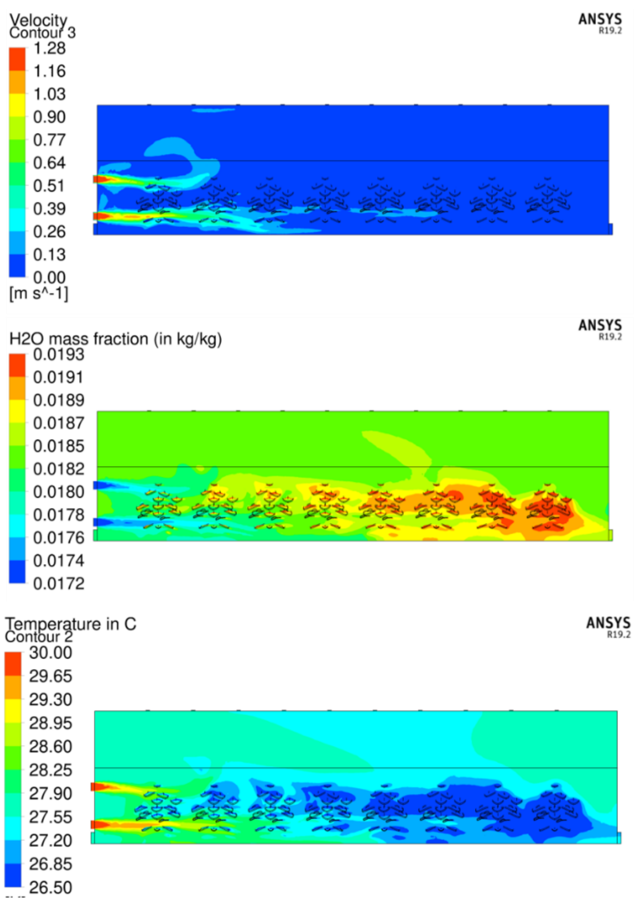

Figure 4a: Case 1: Velocity, water vapour and temperature in a plane situated through the jets

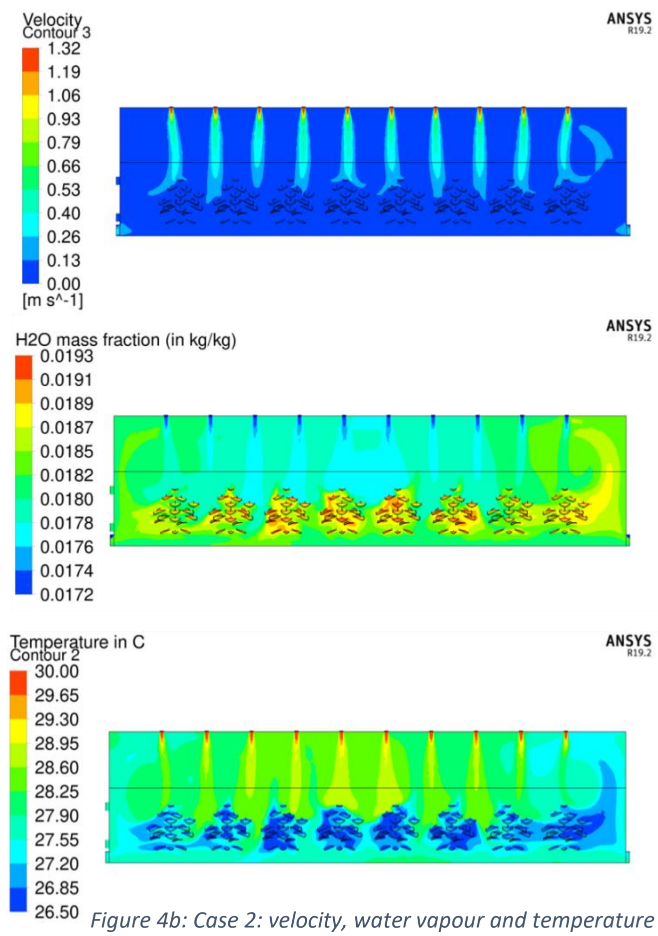

Figure 4b: Case 2: velocity, water vapour and temperature in a plane situated through the jets

Flow uniformity and velocity ranges

To check which is the best configuration, the uniformity of the flow can be investigated. This can be done by looking at the coefficient of variation which is defined as the ratio of the standard deviation to the mean. If this coefficient of variance is high this means that there is a high standard deviation of that variable present and thus a low uniformity. This CV is calculated for the velocity in a plane through the jets, as indicated in Figure 3. A lot of researchers suggest that velocity should be between 0.3m/s and

1m/s for optimal growth of plants, as these higher velocities correspond to a better mass and heat boundary layer, increasing transpiration rates and photosynthetic uptake \cite. The percentage of velocity (PCV) within a certain range is calculated by looking at the velocities in a square box around the plants in the plane through the jets, as indicated in Figure 3. In Table 3, the percentages of velocity (PCV) within a certain range is shown for the four studied cases.

Table 3: Percentages of velocity within a certain range and coefficient of variation

PCV (in %)	v<0.3ms^{-1}	0.3ms^{-1}< v < 1 ms^{-1}	v > 1 ms^{-1}	CV (in %)
Case 1: jets from side	93.4	5.8	0.1	36.0
Case 2: jets from top	98.7	1.3	0	24.6
Case 3: jets from side (double flowrate)	82.3	14.7	3.0	54.1
Case 4: jets from top (double flowrate)	90.2	9.8	0	32.0

Plant transpiration and aerodynamic resistances

Using a special written code in a User Defined Function, UDF, it is possible to calculate the total transpiration of each leaf and the total transpiration of each plant. A higher transpiration value indicates that the undersides of the leaves of the plants are better ventilated. In Figure 5 the transpiration per plant for the four different cases is shown. From the plot, it is clear that in Case 1 and Case 3, where the jets come from the side the first plants are better ventilated and thus have higher transpiration rates.

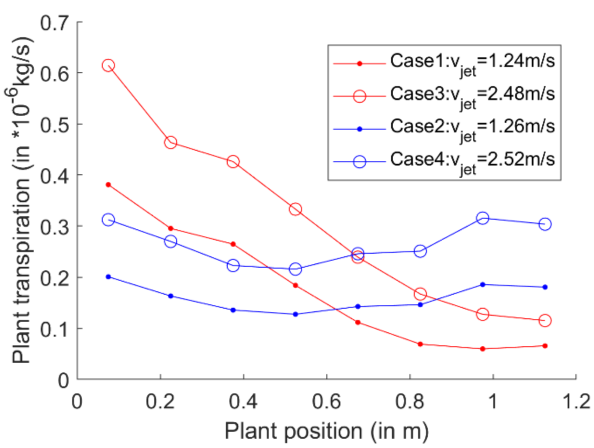

Figure 5: Simulated transpiration for each plant for the four studied cases

Transpiration then drops off towards the end of the plant row and around the fourth plant this transpiration starts to dip below the top – ventilated case. The top-ventilated cases, Case 2 and Case 4 have a more uniform distribution of transpiration rates over the plant row. At both ends of the plant row, the simulated transpiration rates are increasing as these lie close to the pressure outlet in the domain and all the air that comes into the domain is forced to go to the outlets. This air is also less saturated with water vapour, as can be seen in Figure 4b. Furthermore, it is clear that by increasing the air flow rate, Case 3 and Case 4, transpiration also increases but not linearly however.

By multiplying each individual leaf transpiration, ET_{leaf} in kg/s, with the latent heat of evaporation of water, L_v in J/kg and dividing this with the individual leaf area, A_{leaf} in m^2 a latent transpiration flux, q_{lat} in W/m^2 can be defined, see Equation 3.

Equation 3: latent heat flux

$$q_{lat} = L_v \frac{Tr_{leaf}}{A_{leaf}}$$

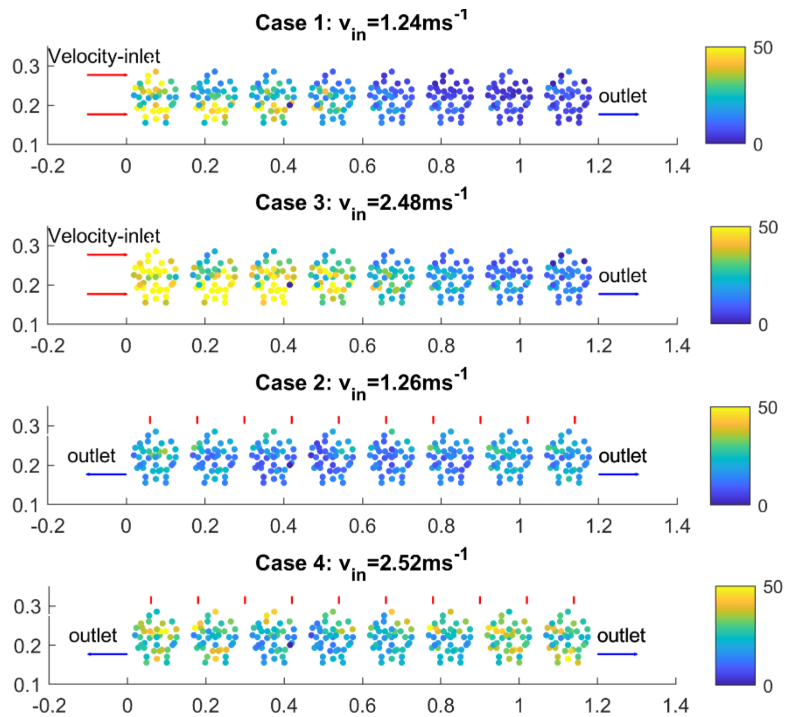

Figure 6: Distribution of latent heat fluxes for each leaf

This latent heat flux can be compared to incoming radiation values and could be inputted as a boundary condition in further simulations. This latent heat flux is also useful to compare the different ventilation systems. In Figure 6, the distribution of these latent fluxes for the four studied cases is shown. Once again, it is clear that the side way ventilated cases are better in ventilating the first plants, but lack the uniformity of flow. The top ventilated cases have a better uniformity, but overall lower transpiration rates. This is caused by the fact that transpiration is included in the model by putting the lower sides of the leaves at a higher humidity. The side ventilated cases are better at ventilating this underside of the leaves, compared to the top-ventilated jets.

Conclusion

Two different ventilation strategies, ventilation from the side using jets and ventilation from the top using jets has been studied for a realistic plant factory environment. Each ventilation option has two set flowrates, making a total of four studied cases. Plants are modelled using a realistic plant geometry. Transpiration is included by putting the humidity at the underside of the leaf at a fixed value. By calculating the water vapour flux from this surface, the individual water vapour flux of each leaf can be calculated. Increasing the flow rate for both cases corresponds to higher momentum in the canopy domain, and with a reduced uniformity. Ventilating from the top has the advantages that plants are more uniformly ventilated. Simulated plant transpirations are more even as well, and the individual latent heat fluxes of each leaf lie more in the same range. When ventilating the plant from the side, the first two to three plants experience high momentums, and thus also high transpiration rates. More momentum is also present in the canopy domain compared to the top ventilated cases. This method has overall proven to be a good way to analyse different ventilation strategies in a plant factory.

Bibliography

[1] Despommier, Dickson, Farming up the city: the rise of urban vertical farms, Trends in biotechnology, Volume 31, 2013, Pages 388-38

[2] Y. Kitaya and J. Tsuruyama and T. Shibuya and M. Yoshida and M. Kiyota, Effects of air current speed on gas exchange in plant leaves and plant canopies, Advances in Space Research, Volume 31, 2003, Pages 177-182

[3] Ying Zhang and Murat Kacira and Lingling An, A CFD study on improving air flow uniformity in indoor plant factory system, Biosystems Engineering, Volume 147, 2016, Pages 193-205

[4] Hui Fang and Kun Li and Gang Wu and Ruifeng Cheng and Yi Zhang and Qichang Yang, A CFD analysis on improving lettuce canopy airflow distribution in a plant factory considering the crop resistance and LEDs heat dissipation, Biosystems Engineering, Volume 200, 2020, Pages 1-12

[5] Benyamin Naranjani and Zabihollah Najafianashrafi and Christopher Pascual and Ireneo Agulto and Po-Ya Abel Chuang, Computational analysis of the environment in an indoor vertical farming system, International Journal of Heat and Mass Transfer, Volume 186, 2022, Pages 122460

[6] Coussement, Jonas and Steppe, Kathy and Lootens, Peter and Rolď an-Ruiz, Isabel and De Swaef, Tom, A flexible geometric model for leaf shape descriptions with high accuracy, Silva Fennica, Volume 52, 2018