CFD Letters 16, Issue 9 (2024) 81-93

SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

A Simulation Study of Hybrid Solar Drying Chamber for Agriculture Product

Wong Kar Hao[1], Mohd Afzanizam Mohd Rosli [1,*], Jayaprakash Ponnaiyan[1], Safarudin Ghazali Herawan[2], Faridah Hussain[3]

[1] Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[2] Industrial Engineering Department, Faculty of Engineering, Bina Nusantara University, Jakarta, 11430, Indonesia
[3] SIRIM Standards Technology Sdn. Bhd., Seksyen 15, 40200 Shah Alam, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 15 August 2023
Received in revised form 17 September 2023
Accepted 16 October 2023
Available online 30 April 2024

Keywords:
Drying chamber; Computational Fluid Dynamics (CFD) simulation; uniformity distribution

ABSTRACT

Hybrid solar drying chamber is an application that is widely used today for agriculture products because it can promise the hygiene of the product. However, drying chambers nowadays still lack uniformity in drying products within a drying chamber, leading to food wastage and compromised product quality. This study aims to design an innovative hybrid solar drying chamber system and investigate the uniformity of temperature and velocity within the chamber using Computational Fluid Dynamics (CFD). The methodology involves validating the simulation results by comparing them with existing journal data, with a validation error of less than 5%. A new design is proposed after the validation process, considering factors such as tray arrangement and air inlet size. The results show that a tray arrangement with 0.20 m spacing between each tray provides better uniformity in temperature and air velocity distribution compared to other arrangements. Additionally, an inlet size of 0.05 m2 demonstrates the most suitable temperature distribution for drying purposes, falling within the ideal range of 318 K to 343 K. The study showed that the performance of the drying chamber under different operating conditions has consistent temperature distribution and is suitable for uniform drying. Overall, the proposed hybrid solar drying chamber system offers improved temperature control and uniformity for effective drying process.

1. Introduction

Drying is a commonly used process to remove water content from agricultural products. In certain places, solar energy has emerged as a popular choice for drying due to its renewable and abundant nature [1, 2]. Traditional methods like open sun drying have limitations in terms of hygiene and product quality. It is sometimes also limited to weather conditions [3]. Solar drying systems, equipped with solar collectors and drying chambers, provide a cleaner and more hygienic environment for improved dried product quality [4]. Besides, solar energy is clean and safe, it can be captured

* *Corresponding author.*
E-mail address: afzanizam@utem.edu.my (Mohd Afzanizam Mohd Rosli)

https://doi.org/10.37934/cfdl.16.9.8193

anywhere and provide energy where other sources cannot. It makes the environment greener and cleaner [5].

The concept of solar dryers started in the 1980s, and they have been involved in the agriculture industry. However, there remains a growing demand for their widespread adoption, especially in developing countries, to be widely used in agricultural production [6]. Currently, people have started investigating the use of secondary heating sources in solar dryers as they can enable the control of drying parameters, extend the drying time, and help overcome the intermittent nature of solar energy. As a result, various hybrid solar dryers using electrical heaters, biomass, and gas burners have been designed and developed to dry different agricultural products [7].

Computational Fluid Dynamics (CFD) is a numerical analysis method used to simulate fluid flows and solve problems related to drying chambers [8]. CFD is particularly useful in predicting air velocity and temperature distributions within the drying chamber [9, 10]. It helps optimize drying performance by comparing simulation results with actual drying experiments [11, 12]. It saves costs and brings convenience to the user.

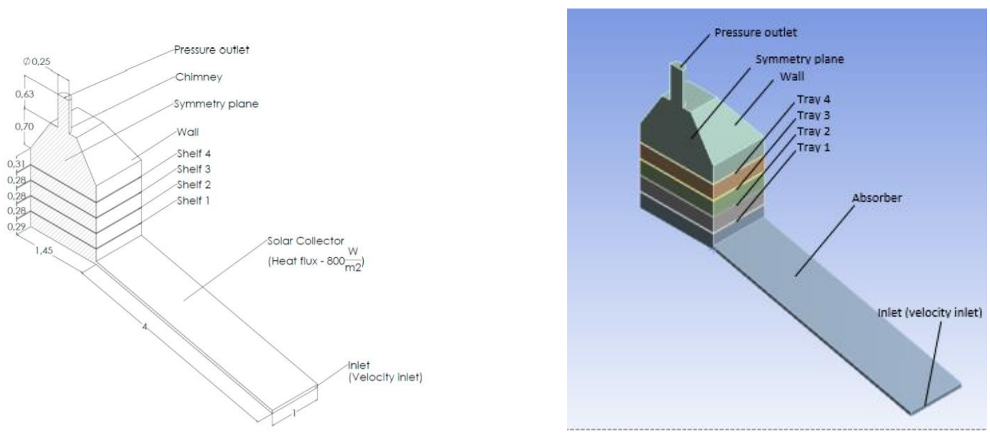
Several parameters significantly affect the drying rate: temperature, air velocity, airflow, and relative humidity/moisture content. Temperature influences evaporation rate and water vapour capacity, while air velocity and flow impact drying efficiency [13]. Uniformity of airflow and temperature distribution is crucial for homogeneous drying [14, 15]. Relative humidity and moisture content determine drying effectiveness, with lower humidity and higher temperatures promoting faster drying [16, 17]. Managing these parameters is essential for optimizing the drying process. In this paper, the focus is the temperature distribution in the drying chamber. In addition, validation is done with the results from Petros *et al.,* [18] and thus proposes a new design.

The objective of this study is to design an innovative hybrid solar drying chamber system and to investigate the uniformity of temperature and air velocity of the drying chamber using the Computational Fluid Dynamics software ANSYS Fluent.

2. Methodology

2.1 Computational Fluid Dynamic Modelling

The drying chamber was designed to consist of four trays or shelves at different levels as a flow domain. The geometry was in symmetrical 3D. Figure 1(a) shows the dimensions of the geometry from the previous study [18], while Figure 1(b) shows the geometry modelled in ANSYS Fluent Design Modeler.

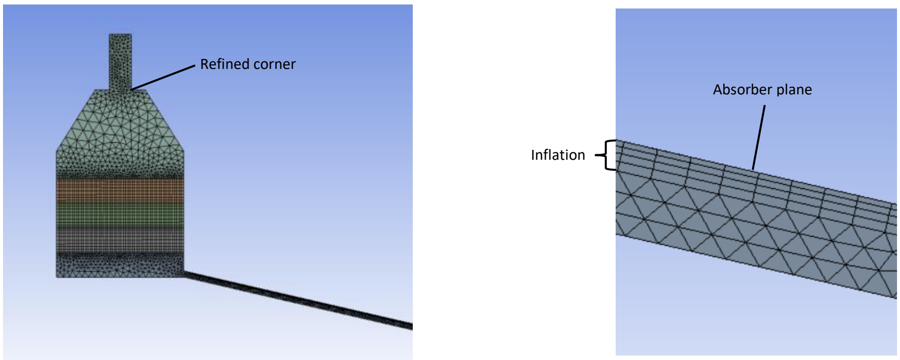


(a) (b)

Fig. 1. Dimension of geometry from (a) reference [18] and (b) Modeled in ANSYS Fluent

2.2 Mesh Generation

In Figure 2, a symmetric 3D flow domain was created using the meshing capabilities of ANSYS Fluent version 2020 R2, resulting in a structured grid representation. To capture details of near-wall flow, the grid was refined at the angle of the corner and inflation was added to the absorber plane. The curvature and proximity were turned on to obtain a more accurate result. The refined meshing of a combination of tetrahedral and multizone was used in the simulation. The meshing method is significant and will affect the accuracy, convergence, and speed of simulation [19].



(a) (b)

Fig. 2. Meshing (a) Side view of meshed geometry and (b) Inflation at absorber plane

2.3 Basic Governing Equations

The simulation involving airflow and temperature distribution within the domain is modelled with mass, momentum, and energy conservation equations [18, 20, 21]. Turbulent flow is modelled in this study. The following equations are solved in CFD:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = 0 \tag{1}$$

Momentum conservation equation:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla.(\vec{\tau}) + \rho \vec{g} + \vec{F} \tag{2}$$

Energy conservation equation:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = \nabla.\left(k_{eff}\nabla T\right) + S_h \tag{3}$$

2.4 Domain Properties and Boundary Conditions

The domain properties and boundary conditions were important in the simulation process to ensure that the result was accurate. The boundary conditions including inlet, outlet, wall, absorber plane and detail in the domain are given in Table 1.

Table 1
The parameters of domain properties and boundary conditions

Domain properties and boundary conditions		Remarks
Inlet	3 m/s velocity	301 K temperature
Outlet	0 gauge pressure	-
Absorber plane	800 W/m^2 heat flux	Default copper properties
Wall	No slip condition	Default aluminum properties
Air domain	-	Default air properties
Air for tray domain	-	Default air properties

2.5 Porous Media Formulation

The food that was to be dried and the trays were approximated to be porous media in the drying chamber. The flow passing through this area is considered a porous zone [22]. The porosity for the tray is assumed to be 0.4 and a relative viscosity of 0.8.

2.6 Simulation

In this study, the ANSYS Fluent 2020 R2 was used to run the numerical simulation. The simulation was performed by solving the conservation Eq. (1), Eq. (2) and Eq. (3) onto the structured grid. The realizable K-epsilon turbulence model, semi-empirical based on transport equation was applied with the scalable wall functions. The simulation was in a transient state with the constant heat flux within the time range of 2 hours of 10 max iterations per time step using the SIMPLE method and hybrid initialization.

2.7 Grid Independence Test

The grid independence test study is performed using six different mesh element sizes coming with six element numbers with acceptable orthogonal quality (see Figure 3). From the table, after 366495 numbers of elements, the velocity output is found to be stable and less than 5%. In the graph, the stable line had been observed when 366495 number of elements. Thus, the dotted line of the element number had been chosen.

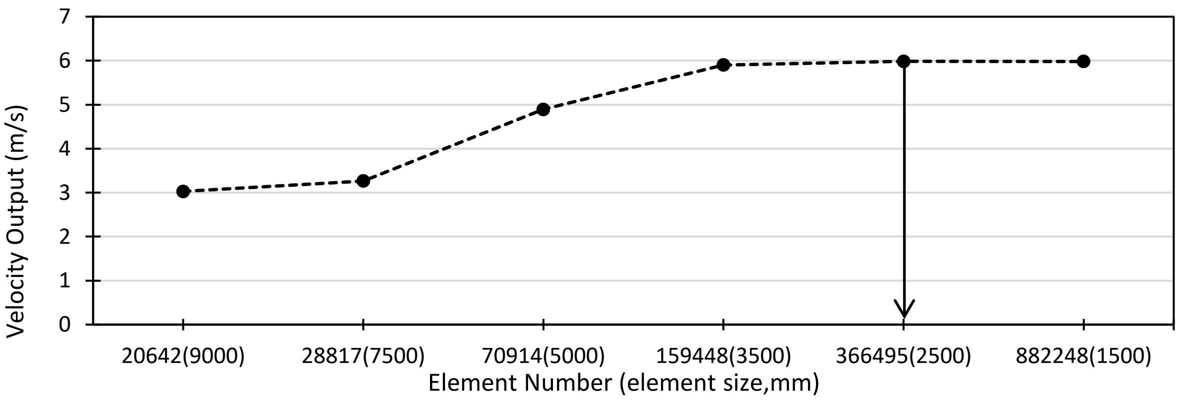
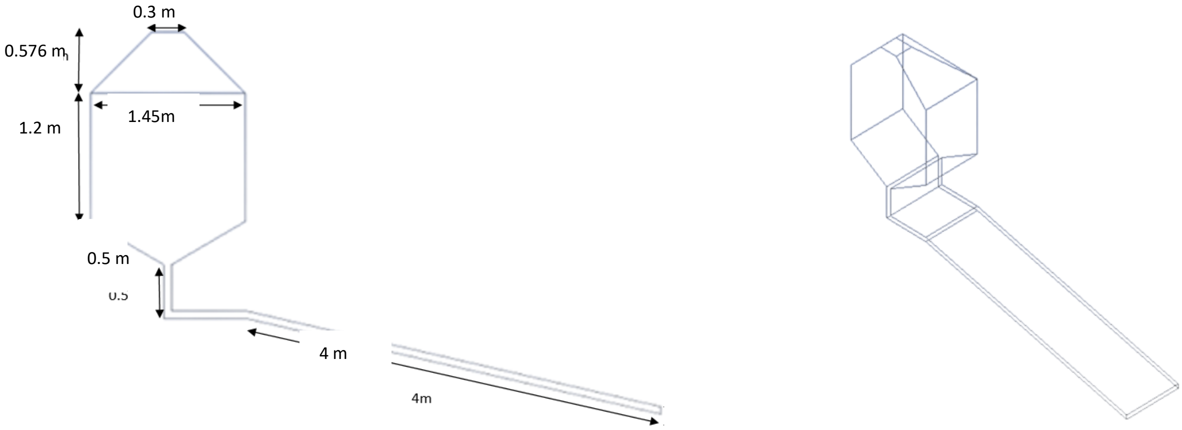


Fig. 3. The graph of velocity output versus element number (element size) for grid independence test

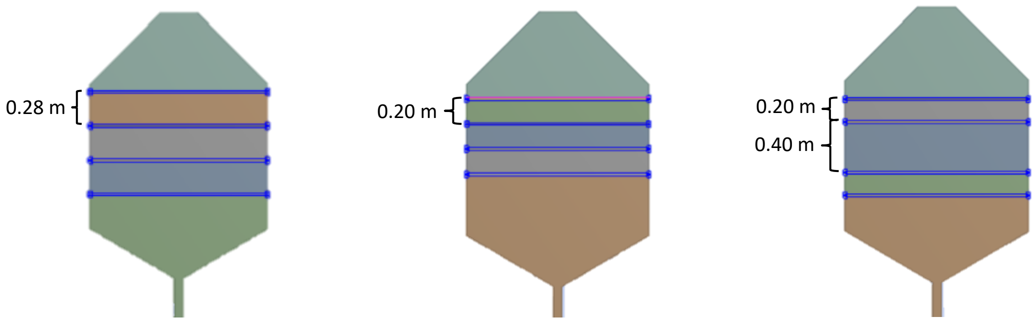
2.8 Proposed New Design

After completing validation, a new design drying chamber will be proposed to promote a better temperature and velocity distribution. The idea came from where heated moist air goes up and leaves the drying chamber through the air outlet at the high end of the dryer [23], so the inlet is designed directly from the bottom in Figure 4. Two parameters were studied. The first parameter is the tray arrangement shown in Figure 5 with three options. The tray arrangement that provided the best result will proceed to the second parameter, air inlet size with three options to investigate their uniformity (see Figure 6). Lastly, the best design from parameter 2 will be continued in the testing in different operating conditions as shown in Table 2.



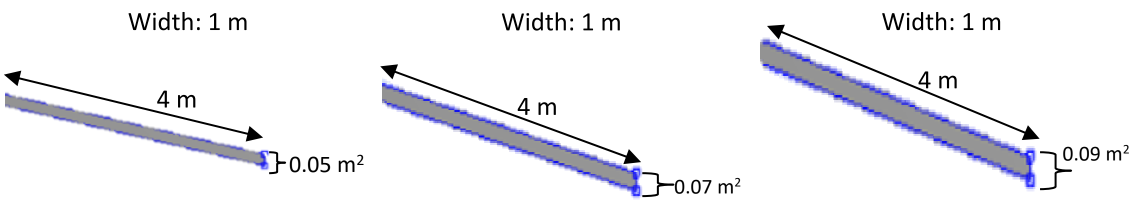
(a) (b)

Fig. 4. Proposed design (a) Dimensions side view and (b) Isometric view



(a) (b) (c)

Fig. 5. Tray gap: (a) 0.28 m, (b) 0.20 m and (c) 0.20 m with the middle big gap of 0.4 m



(a) (b) (c)

Fig. 6. Air inlet size: (a) 0.05 m^2, (b) 0.07 m^2 and (c) 0.09 m^2

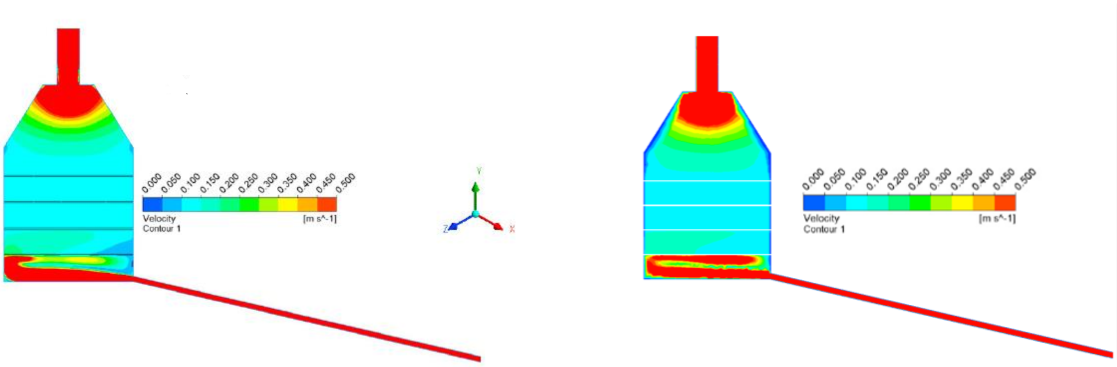
Table 2
Operating conditions

Heat flux (W/m^2)	Air inlet velocity (m/s)
	1
400	2
	3
	1
600	2
	3
	1
800	2
	3

3. Results

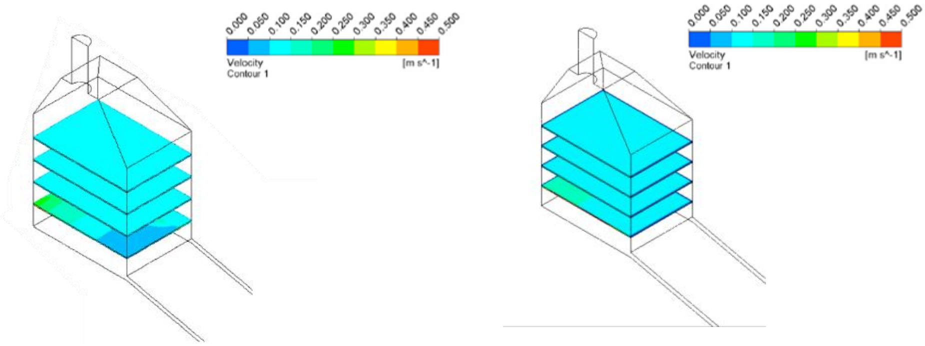
3.1 Validation

To validate the simulation result, a comparison between results from the previous paper [18] and this work CFD simulation was performed. The purpose of this validation is to ensure that the steps and the boundary conditions used for drying simulations are correct. Figure 7 and Figure 8 clearly show that the contour plot of air velocity on the symmetry plane and the four trays of the drying chamber have the same pattern. This can prove that the parameter setting for the simulation is correct. Thus, more accuracy compared to the data such as percentage error had been done in next part.



(a) (b)

Fig. 7. Validated results of contour profile of airflow velocity distribution on the symmetry plane from (a) reference [18] and (b) this works CFD simulation



(a) (b)

Fig. 8. Validated results of contour profile of airflow velocity distribution on the four trays from (a) reference [18] and (b) this work CFD simulation

Figure 9 shows the contour plot of temperature distribution within the drying chamber between the CFD simulation from the previous paper [18] and this work CFD simulation. Four points were tabulated on each tray to get the temperature data. All the temperature data were measured in Kelvin. CFD simulation of this work validation was done by comparing our data with the data obtained from the previous paper [18]. In the validation step, a percentage error within 5% is necessary to convince that the result obtained from the CFD simulation is correct. As shown in Table 3, the range between the difference between the experiment and the CFD simulation on each tray is 0.3 to 2.8, which indicates a 0.1% to 0.9% deviation, and it is within 5% errors.

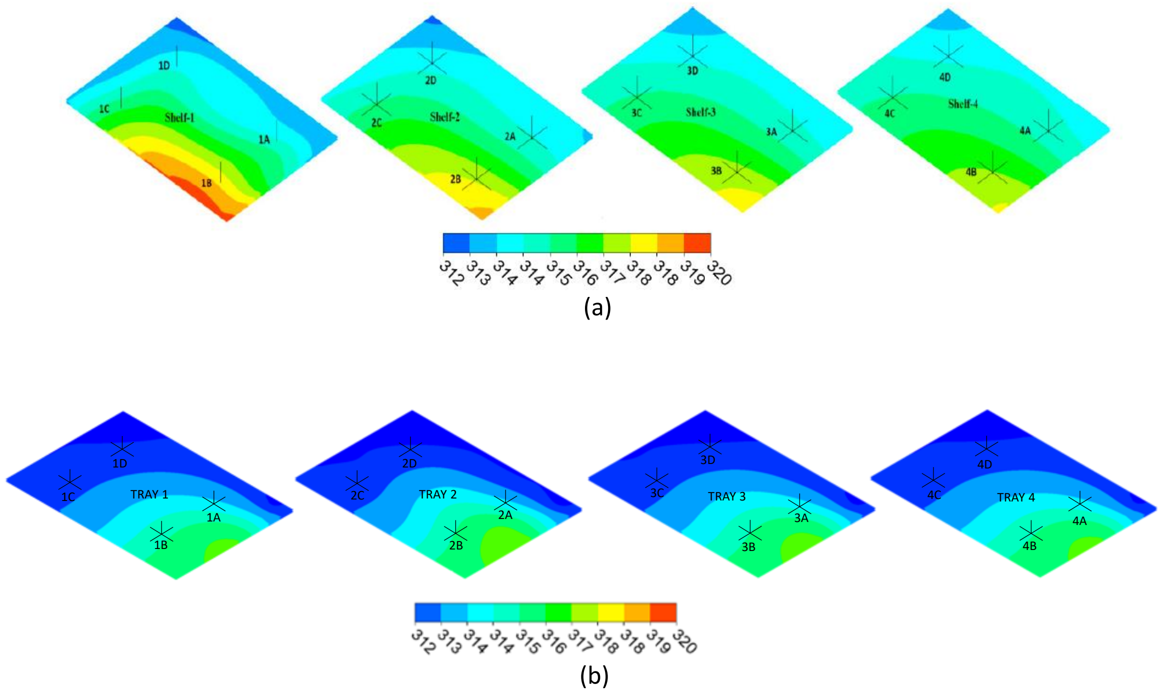


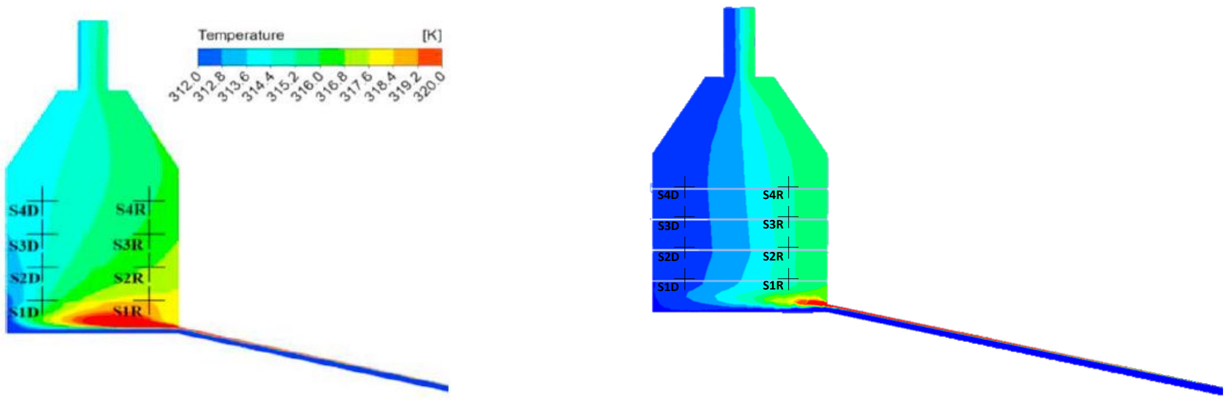
Fig. 9. Validated results of contour profile of temperature distribution on the four trays from (a) reference [18] and (b) this work CFD simulation

Table 3
Temperature data for different points on the different tray

Tray	**1**				**2**			
Location	**1A**	**1B**	**1C**	**1D**	**2A**	**2B**	**2C**	**2D**
Experiment [18]	315.5	314.7	314.1	314.9	317	313.2	312.6	314.4
This work CFD simulation	315.2	315.8	312.3	312.1	314.5	315.6	312.2	312.1
Difference	0.3	1.1	1.8	2.8	2.5	2.4	0.4	2.3
Deviation	0.1	0.3	0.6	0.9	0.8	0.8	0.1	0.7

Tray	**3**				**4**			
Location	**3A**	**3B**	**3C**	**3D**	**4A**	**4B**	**4C**	**4D**
Experiment [18]	314.7	314.2	313.2	313.1	316.3	317	313.3	313.4
This work CFD simulation	314.4	315.5	312.3	312.1	314.3	315.4	312.4	312.1
Difference	0.3	1.3	0.9	1.0	2.0	1.6	0.9	1.3
Deviation	0.1	0.4	0.3	0.3	0.6	0.5	0.3	0.4

Figure 10 shows the comparison of the contour profile of temperature distribution in the symmetry plane between the simulation from the previous paper [18] and this work CFD simulation. It shows that the temperature distribution is slightly different. Hence, a more details comparison via data collected has been done in Table 4. Next, as shown in Table 4, the smaller and largest difference between the experiment and CFD simulation in the symmetry plane is 0.4 and 1.8 respectively which is 0.1% and 0.6%. It means that the accuracy of the simulation has been ensured because they are within 5% errors. Since there are within 5% errors in each tray and symmetry plane, it proves that the steps of the simulation are correct. The difference between the contour from the journal and this work CFD simulation contour is due to the simulation result from the journal has deviation from the experiment.



(a) (b)

Fig. 10. Validated results of contour profile of temperature distribution on the symmetry plane from (a) reference [18] and (b) this work CFD simulation

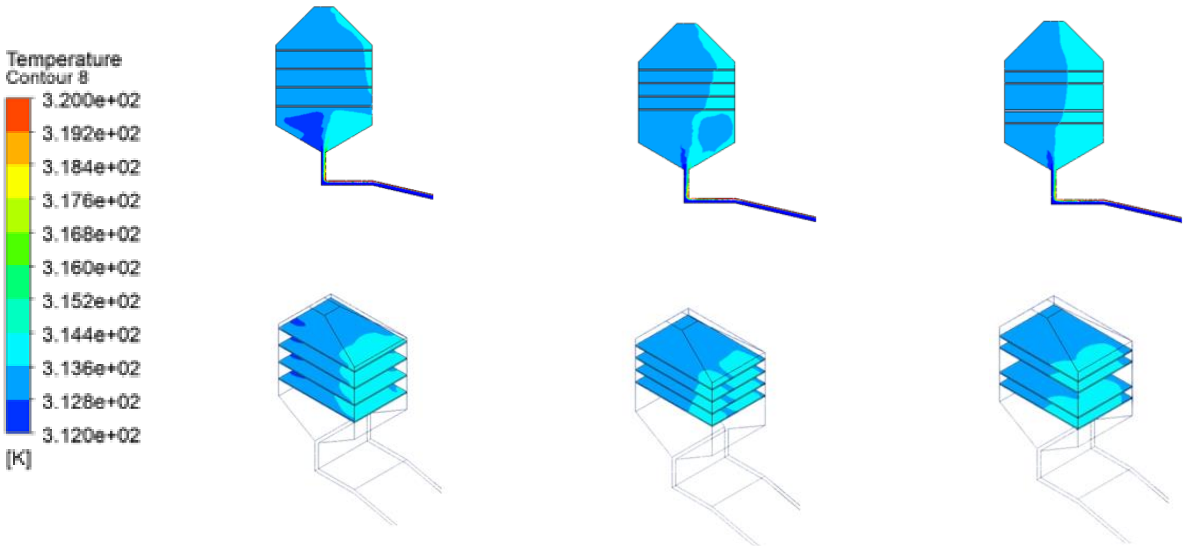
Table 4
Temperature data for different points on the symmetry plane

Side Location	S1R	S1D	S2R	S2D	S3R	S3D	S4R	S4D
Experiment [18]	314.2	314.4	313.9	313.6	312.9	313.9	313.4	313.7
This work simulation	314.7	312.6	314.5	312.5	314.6	312.5	314.7	312.5
Difference	0.4	1.8	0.6	1.1	1.7	1.4	1.3	1.2
Deviation	0.1	0.6	0.2	0.3	0.6	0.4	0.4	0.4

3.2 Parameter 1: Tray Arrangement

For parameter 1, three different tray arrangements were used to study the temperature and velocity distribution. Figure 11 shows the contour profile of temperature distribution for different tray arrangements. As shown in Figure 11, It showed that (b) 0.20 m between each tray provides a better uniform temperature distribution. The data were analysed and show that (b) has a temperature of around 313K on each tray. However, (a) has a temperature range from 312.3 K to 314 K while (c) has a temperature range from 312.8 K to 313.8 K. This shows that the tray arrangement of (b) is much better than the others. There is less fluctuation in (b). Figure 12 shows the details data on each point on each tray.

Uniformity of temperature and velocity distribution can be ensured by keeping away the tray from the inlet because the air velocity is too high at the inlet and poor air flow distribution can cause uneven drying [24]. When trays are placed away from the inlet, it can improve the airflow behaviour inside the chamber [19]. As shown in Figure 13, the velocity distribution is more even at the bottom of the drying chamber when the inlet is located far away from the tray. Hence, the tray arrangement of (b) is selected in parameter 1 and proceeded to parameter 2.



(a) (b) (c)

Fig. 11. Contour profile of temperature distribution in tray gap (a) 0.28 m (b) 0.20 m and (c) 0.20 m with the middle big gap of 0.4 m

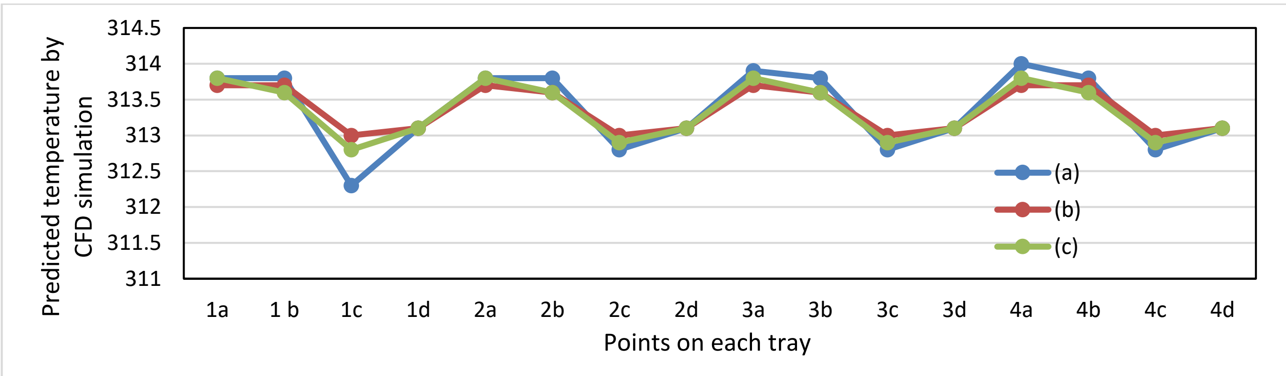
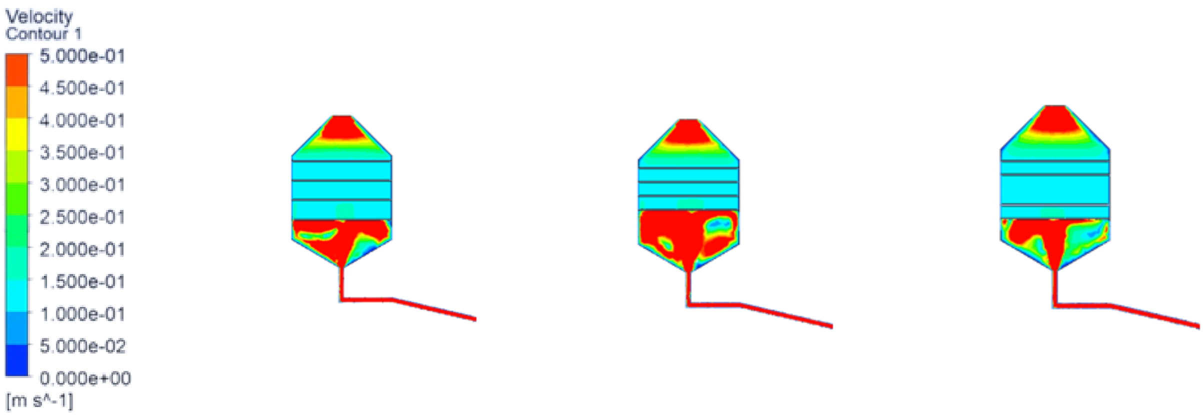


Fig. 12. XY plots of predicted temperature by CFD simulation at points on each tray



(a) (b) (c)

Fig. 13. Contour profile of velocity distribution in tray gap (a) 0.28 m (b) 0.20 m and (c) 0.20 m with the middle big gap of 0.4 m

3.3 Parameter 2: Inlet Size

For parameter 2, (b) from parameter 1 had been taken to proceed with the simulation. An observation from Figure 14, (a) and (c) shows the good temperature distribution at the symmetry plane but (a) has the higher temperature. It shows that 0.05 m^2 has an average temperature of 318 K while 0.07 m^2 has an average temperature of 313 K and 0.09 m^2 has an average temperature of 310 K in the drying chamber. The results show that the distribution of temperature is consistent in each tray. As shown in Figure 15, less fluctuation or more consistency of temperature can be reached when using 0.05 m^2. The temperature becomes lower when the inlet size increases because the mass flow rate increases. When the heat flux is fixed, the increase in mass flow rate can cause a drop in temperature [25].

For drying purposes, the ideal temperature range for drying purposes is between 318 K to 343 K [26]. Hence, parameter 2 of 0.05 m^2 is selected for further study as it meets the minimum temperature.

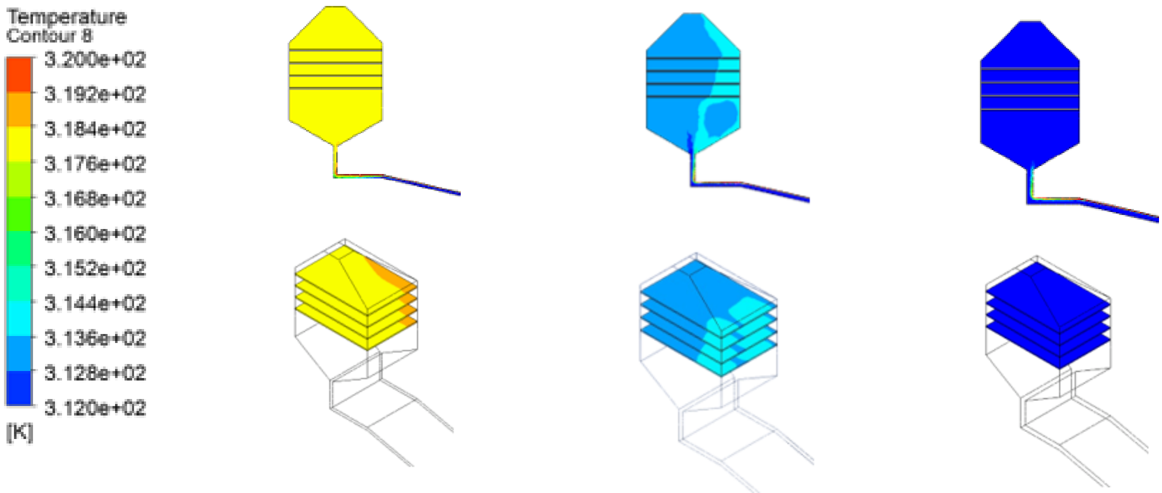


Fig. 14. Contour profile of temperature distribution in air inlet size (a) 0.05 m^2 (b) 0.07 m^2 and (c) 0.09 m^2

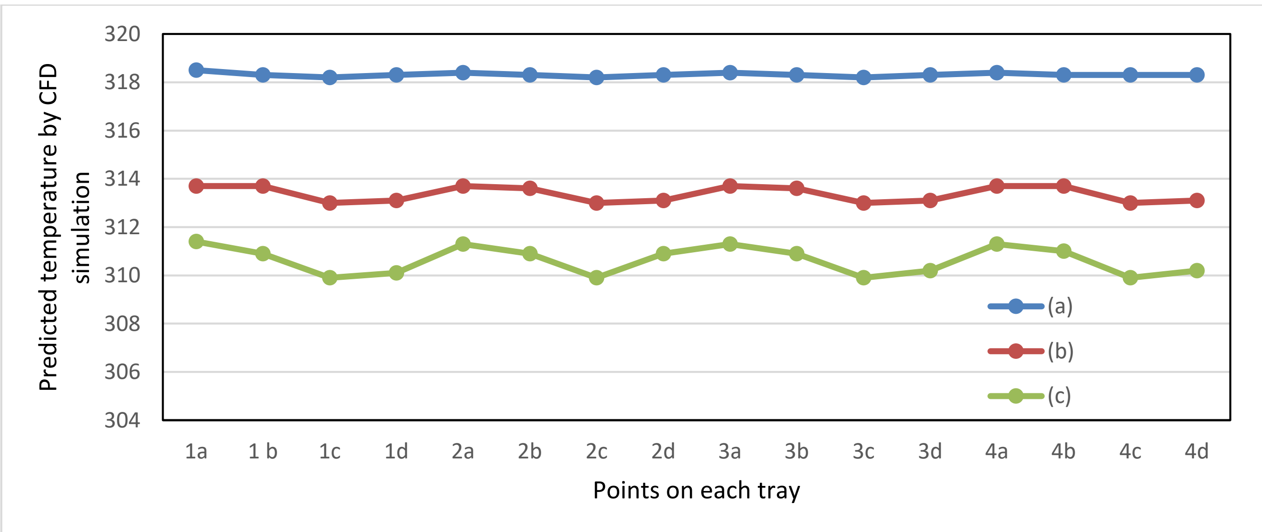


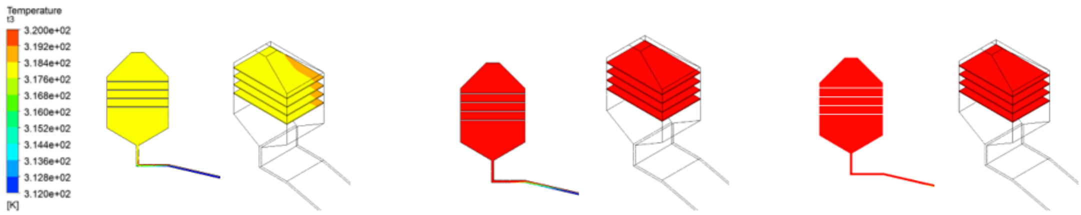
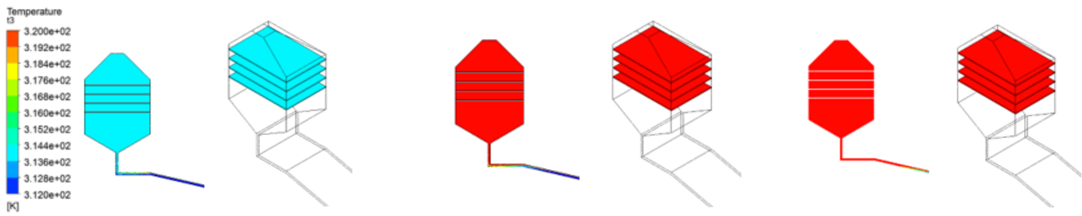
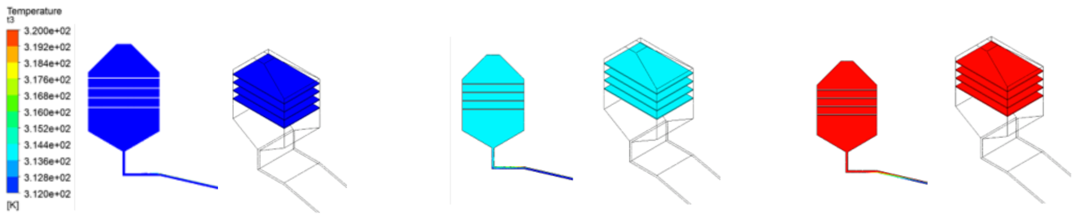
Fig. 15. XY plots of predicted temperature by CFD simulation at points on each tray

3.3 Operating Conditions

In this section, different operating conditions for the design from parameter 2 (a) were simulated to study the temperature and velocity distribution of the drying chamber under various conditions. 800 W/m^2, 600 W/m^2, and 400 W/m^2 of irradiance had been simulated for the design in different inlet velocities which are 3 m/s, 2 m/s, and 1 m/s. As shown in Table 5, the contour profile shows a good temperature distribution as there is not much colour difference in the drying chamber. It also shows that the greater the inlet velocity, the lower the temperature in the drying chamber. Besides, there is less fluctuation and consistency with the temperature in each condition which means the drying chamber performs well in different conditions. It can provide uniform dried products under any conditions.

Since the ideal temperature for drying is within the temperature range of 318 K to 343 K which is claimed by paper [26], it is advised to refer the Table 5 to get the desired temperature for the drying process. Besides, (c), (d), (g), and (h) have an average temperature that is not within the range. Hence, the user can lower or higher the inlet velocity to get a suitable temperature that can be used for drying purposes.

Table 5
Contour profile of temperature distribution in operating conditions

Inlet air velocity Irradiance	3 m/s	2 m/s	1 m/s
800 W/m^2			
	(a)	(b)	(c)
600 W/m^2			
	(d)	(e)	(f)
400 W/m^2			
	(g)	(h)	(i)

4. Conclusions

In summary, From the simulation result, 0.2 m between trays and 0.05 m^2 inlet size shows the best distribution of temperature and air velocity in the new proposed design. It had achieved the objective of designing an innovative hybrid solar drying chamber system. Different operating conditions also give a clear view and understanding of how well the drying chamber can produce uniformity of the product. The results show that consistency or uniform temperature and velocity distribution can be reached by adjusting the tray arrangement and the inlet size. In the end, the temperature and velocity distribution of a drying chamber were investigated. CFD is a good software for predicting temperature distribution and velocity distribution since it saves a lot of time and cost. The result is also convincing because within 5% of the percentage error had been obtained from the validation of the journal.

Acknowledgement
The author would like to thank Fakulti Kejuruteraan Mekanikal, Universiti Teknikal Malaysia Melaka for supporting the project.

References
[1] Sachit, F. A., Noreffendy Tamaldin, M. A. M. Rosli, S. Misha, and A. L. Abdullah. "Current progress on flat-plate water collector design in photovoltaic thermal (PV/T) systems: A Review." *Journal of Advanced Research in Dynamical and Control Systems* 10, no. 4 (2018): 680-89.

[2] Rosli, Mohd Afzanizam Mohd, Danial Shafiq Mohd Zaki, Fatiha Abdul Rahman, Suhaila Sepeai, Nurfaizey Abdul Hamid, and Muhammad Zaid Nawam. "F-chart method for design domestic hot water heating system in Ayer Keroh Melaka." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 56, no. 1 (2019): 59-67.

[3] Rosli, Mohd Afzanizam Mohd, Sit Jing Hiew, Nur Izzati Mohd Azhar, Maidi Saputra, and Sulaiman Ali. "A Simulation Study of Drying Chamber for Marine Product." *International Journal of Integrated Engineering* 13, no. 6 (2021): 62-69. https://doi.org/10.30880/ijie.2021.13.06.005

[4] Singh, Rupanshu, Pranav Salhan, and Anil Kumar. "CFD modelling and simulation of an indirect forced convection solar dryer." In *IOP Conference Series: Earth and Environmental Science*, vol. 795, no. 1, p. 012008. IOP Publishing, 2021. https://doi.org/10.1088/1755-1315/795/1/012008

[5] Ghaderian, J., CS Nor Azwadi, and H. A. Mohammed. "Modelling of energy and exergy analysis for a double-pass solar air heater system." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 16, no. 1 (2015): 15-32.

[6] Yao, Yi, Yoong Xin Pang, Sivakumar Manickam, Edward Lester, Tao Wu, and Cheng Heng Pang. "A review study on recent advances in solar drying: Mechanisms, challenges and perspectives." *Solar Energy Materials and Solar Cells* 248 (2022): 111979. https://doi.org/10.1016/j.solmat.2022.111979

[7] Zoukit, Ahmed, Hicham El Ferouali, Issam Salhi, Said Doubabi, and Naji Abdenouri. "Simulation, design and experimental performance evaluation of an innovative hybrid solar-gas dryer." *Energy* 189 (2019): 116279. https://doi.org/10.1016/j.energy.2019.116279

[8] Sanghi, Achint, RP Kingsly Ambrose, and Dirk Maier. "CFD simulation of corn drying in a natural convection solar dryer." *Drying Technology* 36, no. 7 (2018): 859-870. https://doi.org/10.1080/07373937.2017.1359622

[9] Ram Chourasiya, and Nitin Garg. "Experimental and Simulation Study on Naturalconvection Solar Food Dryer." *International Journal of Research, Science, Technology & Management* 14, no.3 (2019.).

[10] Ambesange, A. I., and S. K. Kusekar. "Analysis of flow through solar dryer duct using CFD." *International Journal of Engineering Development and Research* 5, no. 1 (2017). https://doi.org/10.1016/j.ast.2012.02.006

[11] Mathioulakis, E., V. T. Karathanos, and Vr G. Belessiotis. "Simulation of air movement in a dryer by computational fluid dynamics: application for the drying of fruits." *Journal of food engineering* 36, no. 2 (1998): 183-200. https://doi.org/10.1016/S0260-8774(98)00026-0

[12] Akharume, Felix, Samuel Adeyemi, and Surajudeen Obayopo. "A study on the numerical simulations and experimental validation of a hybrid solar dryer for cocoa." In *2019 ASABE Annual International Meeting*, p. 1. American Society of Agricultural and Biological Engineers, 2019. https://doi.org/10.13031/aim.201900890

[13] Guangbin, Liu, Xu Kaixuan, Yang Qichao, Zhao Yuangyang, and Li Liansheng. "Flow field and drying process analysis of double-layer drying chamber in heat pump dryer." *Applied Thermal Engineering* 209 (2022): 118261. https://doi.org/10.1016/j.applthermaleng.2022.118261

[14] Amanlou, Y., and A. Zomorodian. "Applying CFD for designing a new fruit cabinet dryer." *Journal of food engineering* 101, no. 1 (2010): 8-15. https://doi.org/10.1016/j.jfoodeng.2010.06.001

[15] Moghimi, P., H. Rahimzadeh, and A. Ahmadpour. "Experimental and numerical optimal design of a household solar fruit and vegetable dryer." *Solar Energy* 214 (2021): 575-587. https://doi.org/10.1016/j.solener.2020.12.023

[16] Naing, Than Tun, and Cho Thandar Soe. "Comparative analysis of the performance of cabinet solar dryer and open sun drying for Banana slices." In *IOP Conference Series: Materials Science and Engineering*, vol. 1127, no. 1, p. 012015. IOP Publishing, 2021. https://doi.org/10.1088/1757-899X/1127/1/012015

[17] Seremet, Liliana, Elisabeta Botez, Oana-Viorela Nistor, Doina Georgeta Andronoiu, and Gabriel-Danut Mocanu. "Effect of different drying methods on moisture ratio and rehydration of pumpkin slices." *Food chemistry* 195 (2016): 104-109. https://doi.org/10.1016/j.foodchem.2015.03.125

[18] Demissie, Petros, Mesele Hayelom, Amanuel Kassaye, Asfafaw Hailesilassie, Mekonnen Gebrehiwot, and Maarten Vanierschot. "Design, development and CFD modeling of indirect solar food dryer." *Energy Procedia* 158 (2019): 1128-1134. https://doi.org/10.1016/j.egypro.2019.01.278

[19] Rosli, Mohd Afzanizam Mohd, Yew Wai Loon, Muhammad Zaid Nawam, Suhaimi Misha, Aiman Roslizar, Faridah Hussain, Nurfaizey Abdul Hamid, Zainal Arifin, and Safarudin Gazali Herawan. "Validation Study of Photovoltaic Thermal Nanofluid Based Coolant Using Computational Fluid Dynamics Approach." *CFD Letters* 13, no. 3 (2021): 58-71. https://doi.org/10.37934/cfdl.13.3.5871

[20] Swarup R. Joshi, and Dr. Suresh M. Sawant. “NUMERICAL ANALYSIS OF SOLAR DRYING CHAMBER for AGRICULTURAL PRODUCTS USING CFD.” *International Research Journal of Engineering and Technology* 07, no. 10 (2020): 869-871.

[21] Sileshi, Senay Teshome, Abdulkadir Aman Hassen, and Kamil Dino Adem. "Drying kinetics of dried injera (dirkosh) using a mixed-mode solar dryer." *Cogent Engineering* 8, no. 1 (2021): 1956870. https://doi.org/10.1080/23311916.2021.1956870

[22] Salhi, Mourad, Dounia Chaatouf, Benyounes Raillani, Samir Amraqui, and Ahmed Mezrhab. "Investigating the effect of food trays porosity on the drying process." *Innovative Food Science & Emerging Technologies* 76 (2022): 102939. https://doi.org/10.1016/j.ifset.2022.102939

[23] Pardhi, Chandrakumar B., and Jiwanlal L. Bhagoria. "Development and performance evaluation of mixed-mode solar dryer with forced convection." *International journal of energy and environmental engineering* 4 (2013): 1-8. https://doi.org/10.1186/2251-6832-4-23

[24] Mat, Sohif, Suhaimi Misha, Mohd Hafidz Ruslan, Elias Salleh, and Kamaruzzaman Sopian. "A Study of Drying Uniformity in a New Design of Tray Dryer." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 52, no. 2 (2018): 129-138.

[25] Rahou, M., M. Y. Othman, S. Mat, and A. Ibrahim. "Performance study of a photovoltaic thermal system with an oscillatory flow design." *Journal of solar energy engineering* 136, no. 1 (2014): 011012. https://doi.org/10.1115/1.4024743

[26] Srinivasan, G., D. K. Rabha, and P. J. S. E. Muthukumar. "A review on solar dryers integrated with thermal energy storage units for drying agricultural and food products." *Solar energy* 229 (2021): 22-38. https://doi.org/10.1016/j.solener.2021.07.075