Proceedings of Mechanical Engineering Research Day 2016, pp. 75-76, March 2016

Studying air flow distribution in a tray dryer through CFD simulation

S. Misha[1,2,*], S. Mat[3], M.H. Ruslan[3], E. Salleh[3], K. Sopian[3]

[1)] Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[2)] Centre for Advanced Research on Energy, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[3)] Solar Energy Research Institute, Universiti Kebangsaan Malaysia,
43600 UKM Bangi, Selangor, Malaysia

[*]Corresponding e-mail: suhaimimisha@utem.edu.my

Keywords: Tray dryer; drying simulation

ABSTRACT – Application of tray dryer is widely used in agricultural drying because of its simple design and capability to dry products at high volume. However, the greatest drawback of the tray dryer is uneven drying because of poor airflow distribution in the drying chamber. Implementing the proper design of a tray dryer system may eliminate or reduce non-uniformity of drying and improves drying performance. This study investigates kenaf core drying uniformity in a tray dryer through Computational Fluid Dynamics (CFD) simulation. The result shows that, the higher the average air velocity above the product, the higher the drying rate.

1. INTRODUCTION

Tray dryers are the most widely used dryers for various drying applications because of their simple design and low cost. Generally, a tray dryer consists of several stacks of trays placed in an insulated chamber in which hot air is distributed by a fan or natural flow. The uniformity of airflow distribution over the trays is crucial to obtain uniform product quality. The variation of the final moisture content of the dried product at different tray positions is commonly encountered because of poor airflow distribution [1]. Generally, drying air temperature and velocity significantly affect drying rate [2,3].

Measuring the drying parameters in the drying chamber is expensive, difficult, and time consuming because sensors and data loggers have to be installed in several positions, particularly in a large-scale dryer. Therefore, CFD simulation is used extensively in drying analysis because of its ability to solve systems of differential equations for the conservation of mass, momentum, and energy with the use of advanced numerical methods to predict temperature, velocity, and pressure profiles in the drying chamber

2. METHOD AND SIMULATION

The industrial scale of solar assisted solid desiccant dryer was designed and developed to investigate system performance and drying uniformity in the drying chamber. The experiment setup has been discussed by Misha et al. [4]. The details experimental setup was not discussed in this paper since the focus of this paper to study the airflow distribution in the drying chamber. Kenaf core was used as a sample for drying.

The design of the drying chamber is shown in Figure 1, and includes seven layers of trays, with each layer comprising six trays with dimensions of 64 cm x 92 cm each, for a total of 42 trays. The drying chamber is designed symmetrically from the top view. The sensors are installed only at the right side, assuming that values from the left side are the same, owing to this symmetry. The volume of the drying chamber is 1.7 m (height) x 2 m (width) x 3 m (length). The wall of the dryer system was constructed using 6-cm thick hollow polycarbonate with a hollow space in the middle, 4 cm deep. The top roof is made of glass.

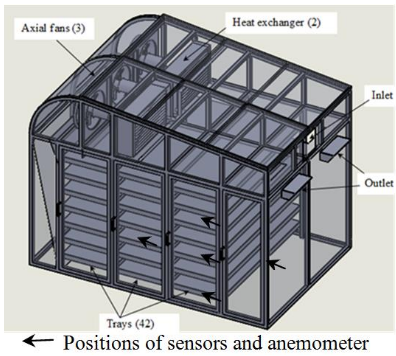

Figure 1 Drying chamber.

The numerical finite volume method used in Fluent 14.0 was used to build a numerical model based on an unstructured 3D mesh using tetrahedral cells. The boundary conditions were set up as follows (Figure 2):

- Inlet 1: The air mass flow rate was 0.58 kg/s (3 m/s), and the air temperature was 44 °C.
- Inlet 2: The air mass flow rate was 0.29 kg/s, and the air temperature was 44 °C.
- Outlet: The gauge pressure was assumed to be equal to 0 at the outlet.
- Porous media: The trays were assumed to be porous with 10% porosity.
- Wall: The heat transfer coefficient of the chamber wall is 4 W/(m^2.K). The environmental temperature is 32 °C, and the temperature at the top roof is 38 °C. The bottom surface was assumed no heat loss. Only half of the drying chamber was analyzed.

© Centre for Advanced Research on Energy

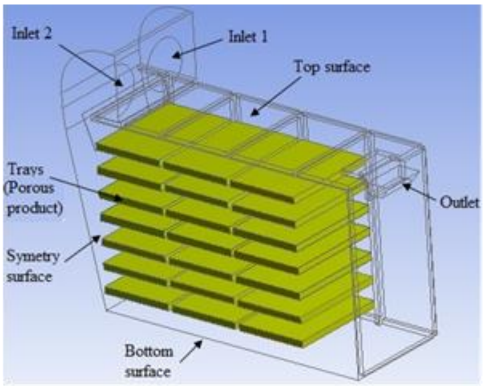

Figure 2 Boundary condition**.**

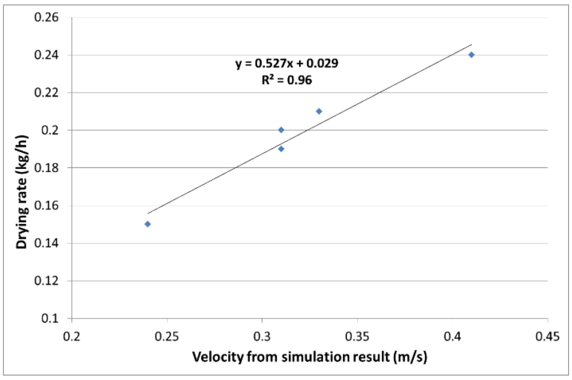

Figure 3 Drying rate against velocity from simulation.

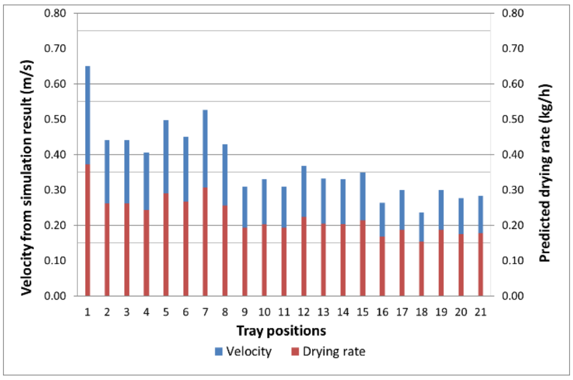

Figure 4 Velocity from simulation and drying rate.

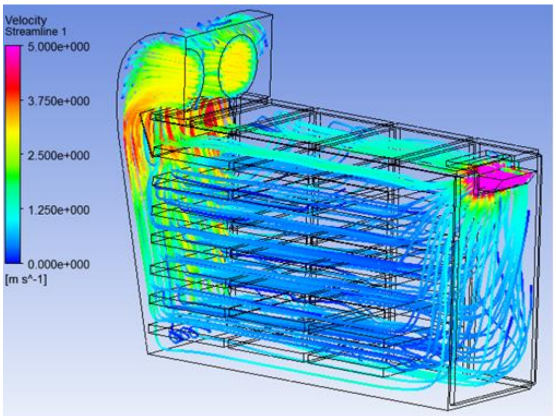

Figure 5 Streamline in the drying chamber.

3. RESULTS AND DISCUSSION

Seven positions in the drying chamber were installed with anemometer to validate the CFD simulation. The simulation values for all points were within the range of anemometer accuracy. Therefore, the simulation results are highly consistent with the experimental data. A plane was created 2 cm above each tray to find the average air velocity for each tray. The velocity at this region was necessary to carry the moisture from the product. In general, as the air velocity increased, the drying rate also increased. The drying rate of the product at trays 4, 9, 11, 13, and 18 was determined. The graph in Figure 3 shows the drying rate and velocity from the simulation data. A strong correlation existed between drying rate and air velocity. The straight line represents the relation between these two parameters. The equation for the straight line is given by:

$$y = 0.527x + 0.029 \qquad (1)$$

Where y is the predicted drying rate and x is the air velocity from the simulation result.

The graph in Figure 4 shows the air velocity from the simulation and the predicted drying rate. The simulation result shows that the 0.24 m/s air velocity above tray 18 was the lowest, and that the drying rate in this region was 0.15 kg/h. The highest air velocity was at tray 1 because of the additional baffle that channels the air to the top tray. Without the baffle, less air was channeled to the top tray and lower air velocity was produced. The simulation without an additional baffle was conducted to predict air flow. The 3D simulation result of the air stream is shown in Figure 5.

4. CONCLUSIONS

CFD simulation was used to predict air flow distribution in the drying chamber by considering the product as porous media. The experimental and simulation data were in excellent agreement. The average air velocity above each tray was predicted through simulation. The drying rate of the product was found to be significantly influenced by the average air velocity above the tray, which carried moisture away from the product. In view of the correlation between the experimental and simulation results, the uniformity of the drying for the products at different levels but at the same column was considered acceptable.

REFERENCES

[1] S. Misha, S. Mat, M.H. Ruslan, K. Sopian and E. Salleh, "Review on the Application of a Tray Dryer System for Agricultural Products," *World Applied Sciences Journal*, vol. 22, no. 3, pp. 424–433, 2013.

[2] F. Pinaga, A. Mulet, A. Berna and M. Borras, "Effect of air flow rate on carrot drying," *Drying Technology*, vol. 5, no. 2, pp. 245–258, 1987.

[3] V.G. Belessiotis, and V.T. Karathanos, "Sun and artificial air drying kinetics of some agricultural products," *Journal of Food Engineering*, vol. 31, no. 1, pp. 35–46, 1997.

[4] S. Misha, S. Mat, M.H. Ruslan, E. Salleh, and K. Sopian, "Performance of a solar assisted solid desiccant dryer for kenaf core fiber drying under low solar radiation," *Solar Energy*, no. 112, pp. 194-204, 2015.