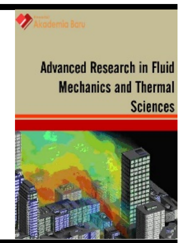
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879




# The Effect of Ventilation Hole Number on Flow Behavior and Heat Transfer of Rotary Drum Dryer

Open Access

Sherly Hanifarianty[1], Thanansak Theppaya[1,2,3], Chayut Nuntadusit[1,2,3], Makatar Wae-Hayee[1,2,3,*]

[1] Energy Technology program, Faculty of Engineering, Prince of Songkla University, 90112 Hatyai, Songkhla, Thailand
[2] Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, 90112 Hatyai, Songkhla, Thailand
[3] Energy Technology Research Center, Faculty of Engineering, Prince of Songkla University, 90112 Hatyai, Songkhla, Thailand

**ARTICLE INFO**

***Article history:***
Received 26 March 2018
Received in revised form 22 May 2018
Accepted 10 June 2018
Available online 14 June 2018

***Keywords:***
Rotary dryer, CFD, Heat transfer, Ventilation hole, Flow behavior

**ABSTRACT**

The effect of ventilation holes on flow behavior and heat transfer of rotary drum dryer were studied by using Computational Fluid Dynamics (CFD). In the experimental apparatus, the drum was heated by direct burning with flame from LPG at the bottom surface of the drum. In steady state condition, the temperature distribution on the surface of the drum was detected by using infrared camera. The average temperature on the cap surface was 80$^{o}$C, the drum surface was 130$^{o}$C, and the temperature inside rotary drum was 120$^{o}$C. In order to maintain the air temperature inside the drum at 120$^{o}$C, the number of ventilation holes was varied by using CFD. In the 3-D numerical model of the drum including ambient air was simulated in transient. The ventilation holes were drilled on the drum caps, and the number of the hole on each cap was varied at 6, 8, 12, and 18 holes. The results showed that the air temperature inside the drum at the condition of ≥12 holes was the highest with the shortest time. The longest time of 1,400 seconds and 1,600 seconds were observed to be uniform temperature and uniform velocity flow in this research.



## 1. Introduction

Drying is the process removal of moisture by transferring heat that can change the quality of product [1]. In this process, heat is transferred from the hot dry air to the product, before evaporating the moisture of the surface material. There are two sequential processes when moisture material undergoes thermal drying [1]. For the first process, removal of water as vapour from the surface material depends on outside condition, temperature, air moisture and flow, and surface area exposed. For the second process is removal of moisture as function of natural physical from solid temperature. Inside the drying operation, this process can be limited by drying rate factor, although this happen simultaneously throughout the drying cycle.

* *Corresponding author.*
*E-mail address: wmakatar@eng.psu.ac.th (Makatar Wae-Hayee)*

Nowadays, rotary drum drying was being developed because of the uniformity of material being dried inside the rotary drum [2-4]. Some of experts have developed to study about flow behaviour for continuous rotary drum [5-8]. Alonso *et al.,* [9] also reported that rotary drum in solar thermal application give positive effects such as good mixing and homogeneous distribution of temperature inside the chamber.

Previous study has been conducted by Delele *et al.,* [10] about rotary drum speed, feed rate and drum angle that showed strong effect of particle and fluid flow behaviors inside rotary drum. Santos *et al.,* [11] also investigate particle size and density differences that could affect physically in the rotary drum. Machado *et al.,* [12] reported rotary drum with different rotation speed, while Liu *et al.,* [13] investigated liquid gas particle in a rotary drum that showed not clear effect on gas improvement and particle mixing in the process.

In rotary drum drying technique, simulation particle need to be considered such as moisture content of solid, air distribution and temperature inside the drum. Some researchers has reported about simulation particle, Geng *et al.,* [14] and Geng *et al.,* [15] reported about particle transport the axial direction of drum and transverse direction of drum. Gu *et al.,* [16] and Gu *et al.,* [17] investigated about particle and gas flow behaviour in rotary drum. Geng *et al.,* [18] studied about particle motion of rotary drum dyer. Karunarathne *et al.,* [19] and Santos *et al.,* [11] reported about particle behaviour with different size and density differences for rotary drum dryer. Furthermore, Nafsun *et al.,* [20] also reported about particle size to thermal mixing behavior, therefore no reported about design rotary drum dryer with ventilation hole as wet warm air removal.

In this study, a rotary drum dryer was designed with varying ventilation holes. The model of rotary drum dryer is shown in Figure 1. Burners were located at the bottom of the drum for heating. The wet warm air passed through the ventilation hole. In the previous of our work, the study of laboratory experiment palm fruit drying by using a newly-designed of rotary drum dryer have been done [21]. As well as, preliminary study of flow behaviour and heat transfer of the drum dryer was simulated by using CFD [22]. CFD was used by many fields for example in industrial device [23], fossil fuel [24], solar energy [25], gas turbine [26], and some in biomedical field [27]. In this work, the effect of ventilation hole number on flow behaviour and heat transfer of rotary drum dryer were studied by using Computational Fluid Dynamics (CFD).

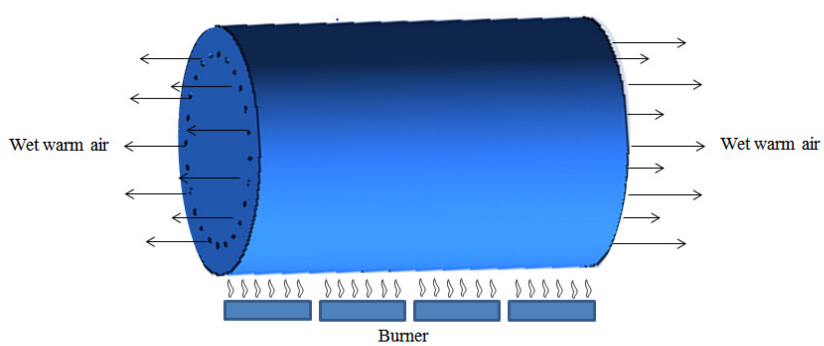



**Fig. 1.** Newly-designed of rotary drum dryer

## 2. Methodology

### *2.1 Rotary Drum Dryer*

The design of rotary drum dryer was built up with metal sheet with thickness of 3 mm. For the dimension was fabricated with 90 cm in length and 57.5 cm in diameter. The rotary drum has roller that was rotated as function with gear and electrical motor. Later on, the rotary drum was heated by

4 burners at the bottom of the drum using LPG as a fuel source. The details of the rotary drum were in our previous work [21].

As shown in Figure 2 and Figure 3, the temperature on the surface of rotary drum and on the cap were captured with steady state condition using infrared camera. The drying process temperature in rotary drum was averaged as 80$^o$C of cap temperature, 130$^o$C of drum surface temperature and 120$^o$C of the air temperature at the center of rotary drum. At this point, the drying process of rotary drum dryer was heated with LPG by applying direct contact to the bottom of the drum. The rotation speed of the rotary drum was slowly at 1.68 rev/hour, 4.14 rev/hour, and 8.34 rev/hour.

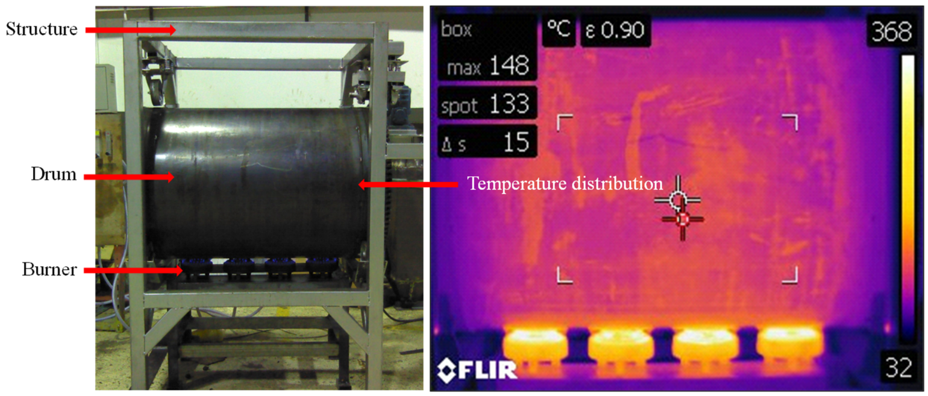



**Fig. 2.** Left: Photo of rotary drum, and Right: Temperature distribution of rotary drum

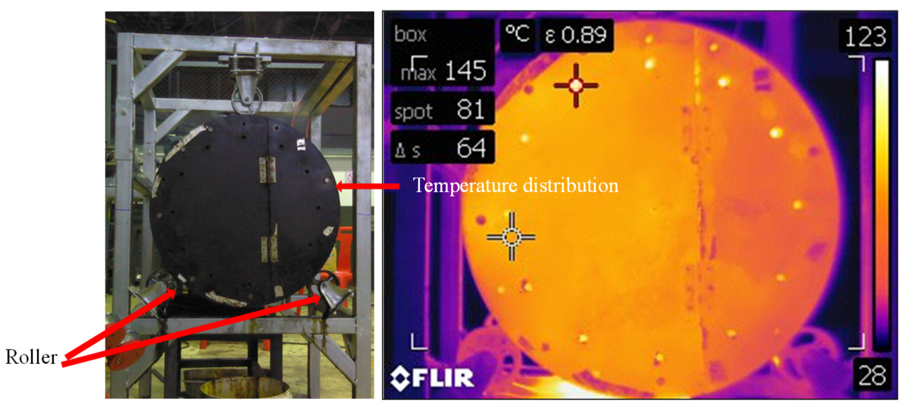



**Fig. 3.** Left: Photo of surface cap, and Right: Temperature distribution of surface cap

### *2.2 Design of Simulation and Grid*

In this simulation work, ANSYS Fluent version 15.0 was used to simulate the flow characteristic and heat transfer of rotary drum dryer. A 3−D with finite volume method was applied to solve governing equation as boundary condition of rotary drum. In order to minimize the calculation task, the size of numerical model was created with decreasing proportionally 10 times smaller than the real drum. The 3-D numerical model of the drum including ambient air is shown in Figure 4.

The grid system of the numerical domain is shown in Figure 5. In cutting section was observed and appeared internal grid system. The geometries of elements were rectangular shape. In order to evaluate saturated element number, the variation of element number at 0.27, 0.71, 1.20, and 1.57 million elements were examined. The flow velocity of hot air discharging from ventilation hole at

*Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*
Volume 46, Issue 1 (2018) 62-72

variant element number is shown in Figure 6. It showed that the trend of velocity was higher when the element number became larger. For the case of 1.20 and 1.57 million elements, the trend of velocity was almost the same. Therefore, 1.2 million cases were being used to be applied in this simulation.

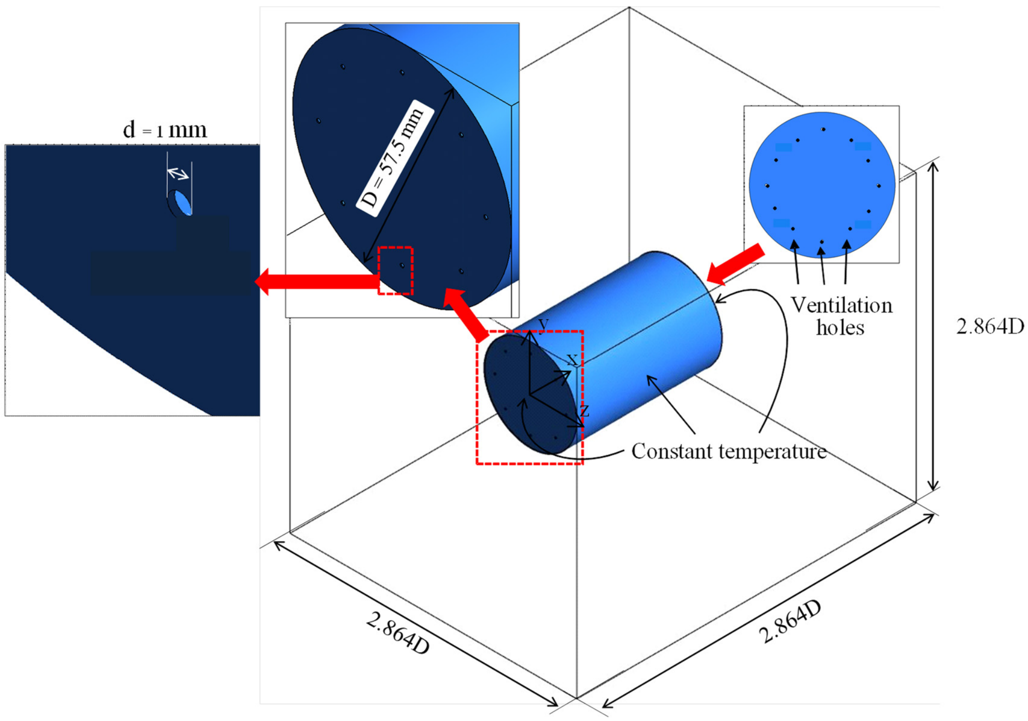



**Fig. 4.** Rotary drum dryer model and surrounding (minimizing proportionally 10 times respecting the actual size of experimental drum)

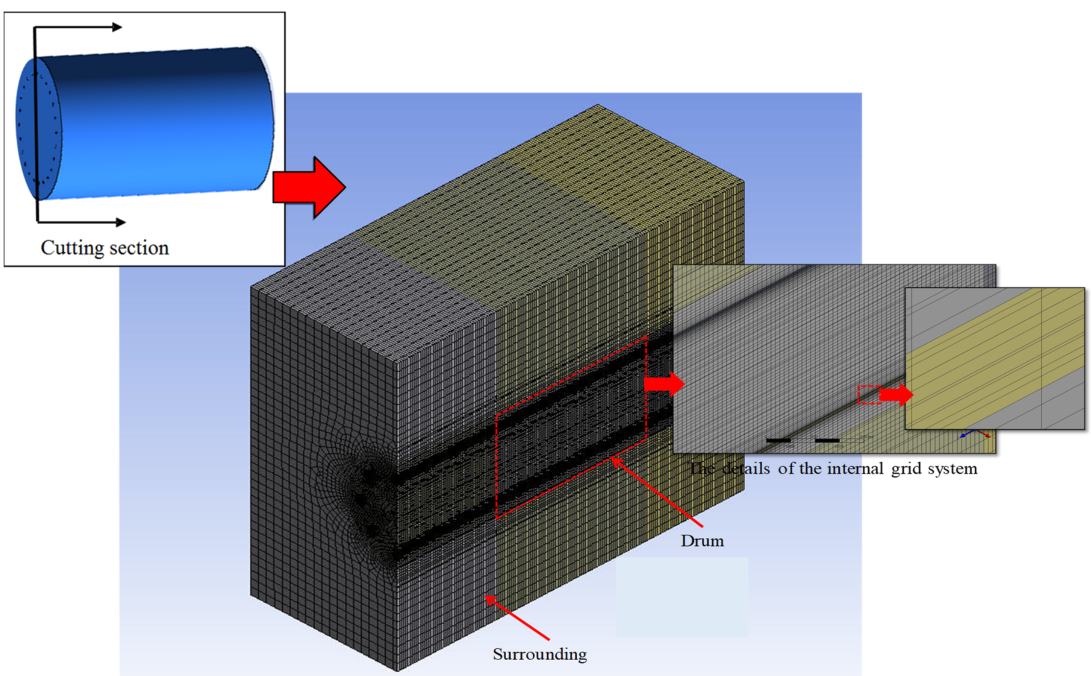



**Fig. 5.** Internal grid of the simulation domain

For the boundary condition of numerical domain was applied with similarity of experimental rotary drum. The details of the boundary conditions were shown in Table 1.

**Table 1**
Boundary condition of rotary drum dryer

| Conditions | Setting |
|---|---|
| The temperature of drum dryer | 130 °C |
| The temperature of drum dryer cap | 80 °C |
| The temperature of environment | 29 °C |
| Ventilation holes | Pressure outlet |
| Outer surface of surrounding air | Pressure outlet |

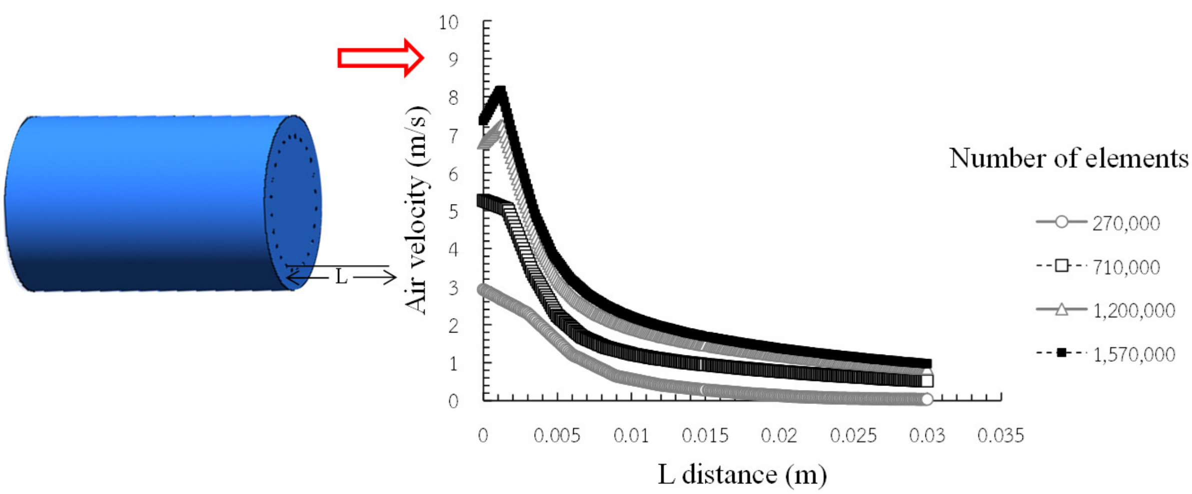



**Fig. 6.** The effect of element numbers on air velocity of the centre of ventilation holes to the surrounding (L is the distance from ventilation hole to surrounding air)

### *2.3 Simulation Assumption*

To simplify numerical simulation, the empty drum was heated without drying material, and the drum was no rotation due to very slow rotating speed (1.68 rev/hour) for the test case (the case of capturing temperature on the surface). The heat from high temperature on the surface of the drum and the caps transferred to the air inside the drum. Therefore, the calculation was considered under unsteady state. The gravity effect was applied, and incompressible flow was considered.

### *2.4 Calculation Method*

The solution in this simulation was evaluated with SIMPLE algorithm and a second order upwind for all spatial discretization. It was considered in transient condition, and then operated until the temperature at the center of drum become consistent. The solution was considered to be converged when the normalized residual of the algebraic equations were less than a prescribe value of 1.0 x $10^{-4}$.

## 3. Results and Discussions

The criteria for identifying optimal hole numbers for designing the rotary drum dryer, the temperature of air at the center of the drum were considered for getting high temperature in the shortest heating period. The air temperature at the center of rotary drum varying with hole simulation is shown in Figure 7. The increasing of the air temperature at the center of drum depended on time in the range of <1,200 seconds. The air temperature at the center of drum became steady state at time <1,200 seconds, approximately. At 6 holes, the increasing of air temperature at the center of drum was faster than those other cases, but at steady state (time <1,200 seconds), its temperature was lower than those other cases. From the figure, it showed that the temperatures at the center of drum for the case of 12 and 18 holes were the highest. As a result of experimental case of 18 holes was the temperature with time at the centre of rotary drum. The temperature was rapidly increased until <2,400 seconds of drying period. The temperature distribution at the centre of drum become almost steady after <2,400 seconds of drying period. Thus, the period of the temperature at the drum centre becoming steady is longer compared with other CFD cases. It was due to effect of heat losses from radiation and natural convection in the experiment.

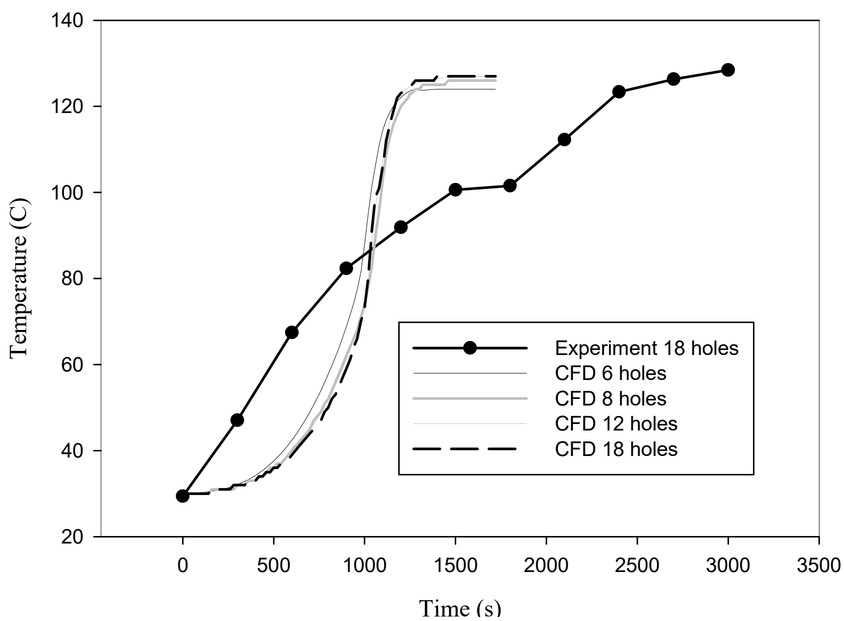



**Fig. 7.** Temperature with time at the centre of rotary drum varying with time

The temperature along the centerline at the middle drum in various ventilation hole are shown in Figure 8. At first, the lower temperature in rotary drum was observed with 6 ventilation hole number. The higher temperature of middle drum increased when the number of ventilation hole is larger. At this point, the temperature along the centerline at the middle drum for the case of 12 and 18 holes were the highest.

Velocity vectors and temperature distributions on X-Y plane of rotary drum dryer simulation which varied with time in period of 200 – 1,600 seconds are shown in Figure 9. At 200 seconds (Figure 9(a)), the temperature was increased on time started from the edge of the drum, and the velocity vectors discharged from the ventilation holes to surrounding due to expansion effect from the increasing air temperature. When time became 400 seconds (Figure 9(b)), the air temperature near the drum edges was higher and the velocity vectors discharging from the holes were also larger. At 600 seconds (Figure 9(c)), the area of air temperature near the drum edge became larger than those

*Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*
Volume 46, Issue 1 (2018) 62-72

former cases, but the air temperature inside the drum still lower and comparable to those former cases.

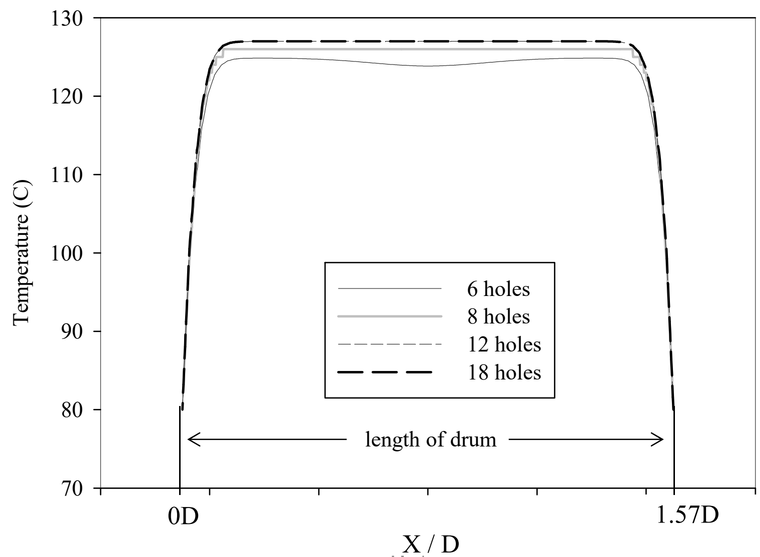
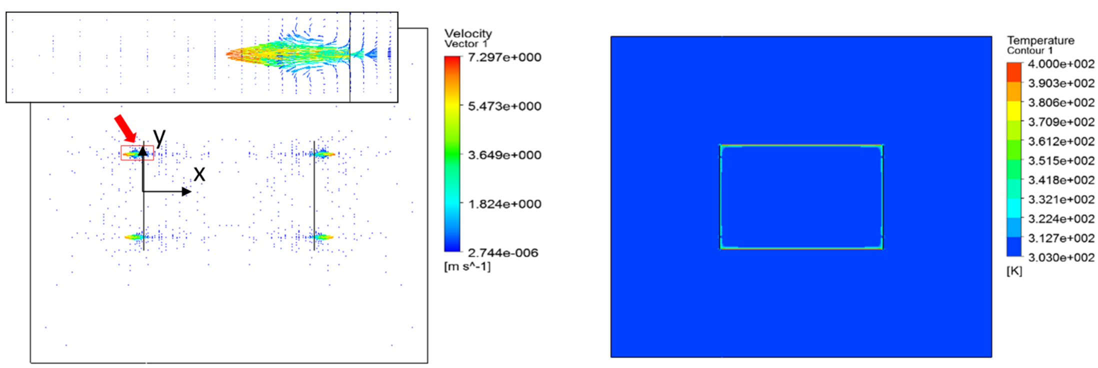
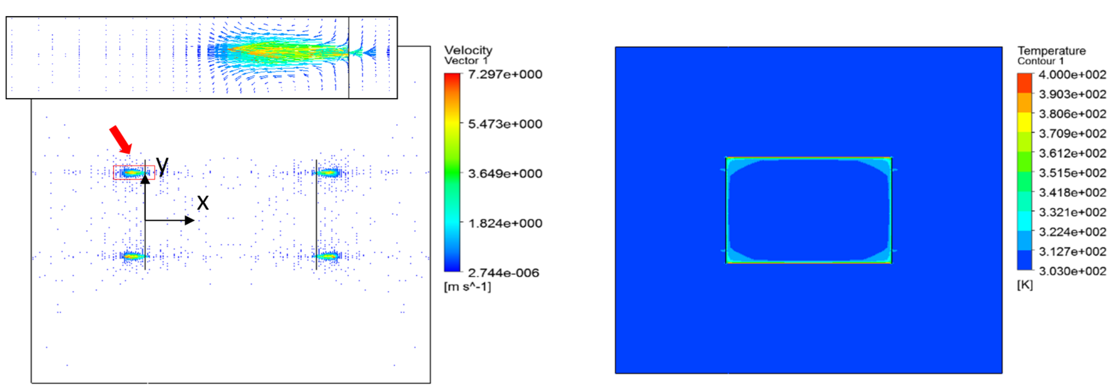


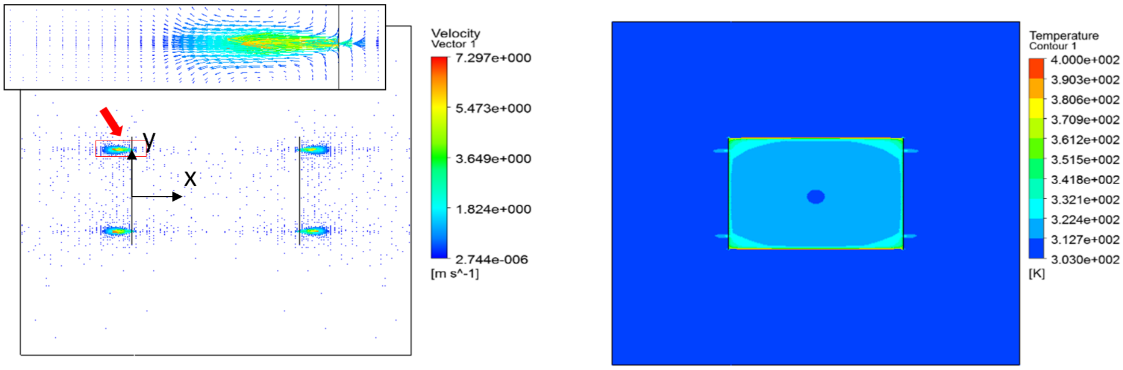

**Fig. 8.** Temperature along the centreline at the middle drum



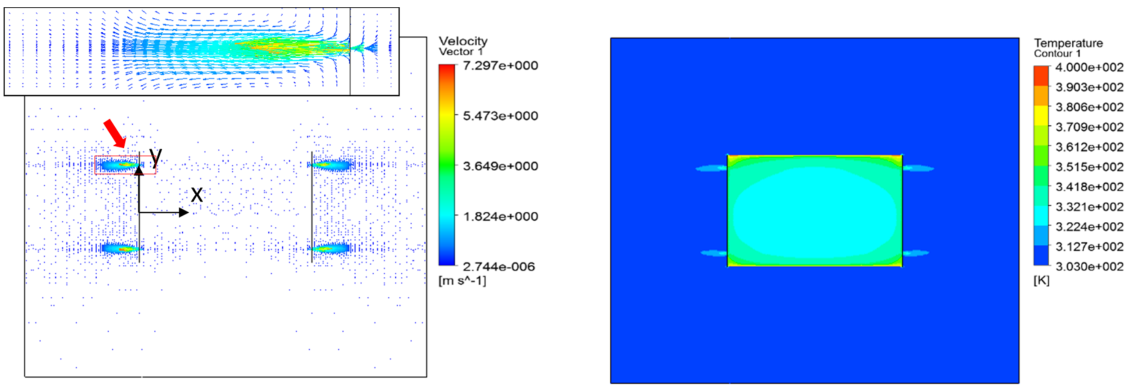

(a) At 200 seconds



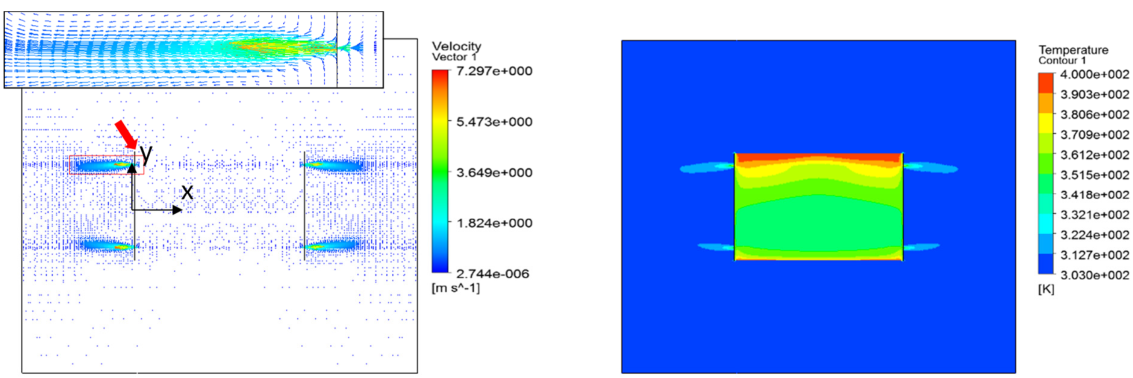

(b) At 400 seconds



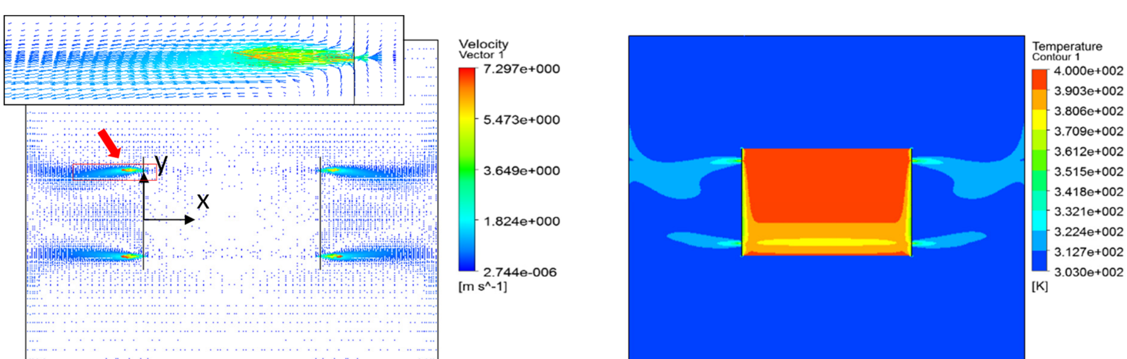

(c) At 600 seconds



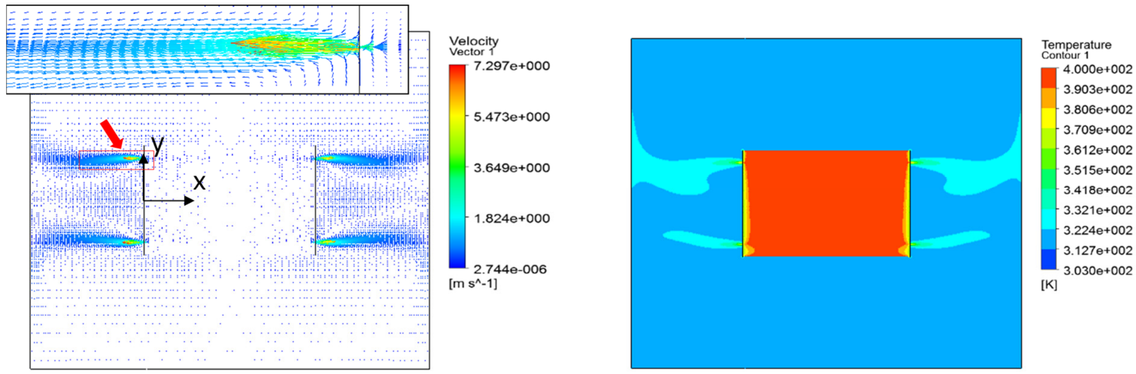

(d) At 800 seconds



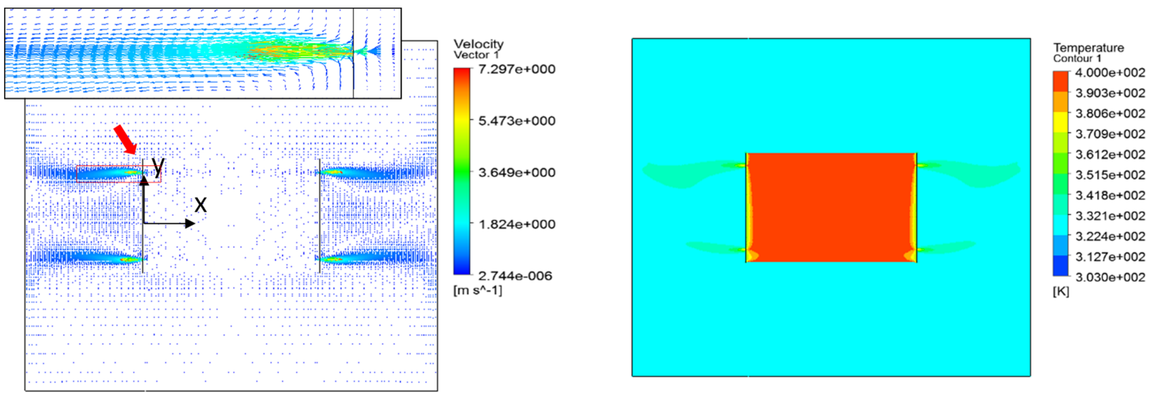

(e) At 1000 seconds




(f) At 1200 seconds




(g) At 1400 seconds




(h) At 1600 seconds

**Fig. 9.** The simulation results on X-Y plane, Left: Velocity vectors, and Right: Temperature distributions (continued)

At 1,000 seconds (Figure 9(e)), asymmetry temperature distributions were appeared due to the effect of natural heat convection [28]. The high temperature was found in upper section of the drum, and then the lower temperature was found in below section of the drum. The different temperature along the height of the drum was directly influenced to the air expansion of the drum which can be identified from discharging air velocity at upper ventilation hole was higher than that of lower ventilation hole (Figure 9(e), left). Afterwards, at 1,200 seconds (Figure 9(f)), the area of high temperature distribution inside the drum became larger, but the asymmetry temperature distributions were found. For 1,400 seconds (Figure 9(g)), and 1,600 seconds (Figure 9(h)), showed the uniform temperature distributions for the whole inside the drum. On the average, for both 1,400 and 1,600 seconds were shown to have the highest velocity flow discharging from ventilation holes.

## 4. Conclusions

In this work, a rotary drum dryer was designed and fabricated by varying ventilation hole numbers at 6, 8, 12, and 18 holes which was drilled on the drum caps. In order to observe hot air velocity discharging from the ventilation holes and air temperature distributions inside the drum, CFD technique was adopted under transient simulation. The temperature distributions on the drum surface and the drum caps of the experimental apparatus which were captured using infrared thermal imager were specified in the boundary conditions of numerical domain. The results showed that the air temperature inside the drum increased depending on the time. Hot air inside the drum discharged to surrounding due to expansion effect from the increasing of air temperature. The air temperature at the drum center in condition of ≥12 holes was the highest with the shortest time. At 1,400 and 1,600 seconds for the case of 12 ventilation holes, temperature distributions were more uniform that the former times. These results suggested that velocity flow and temperature distributions of different ventilation holes and temperature variations have to be studied further.

**Acknowledgement**
This research grant was supported by Energy Policy and Planning Office (EPPO), Ministry of Energy, Thailand, grant No. EE-PSU-59-02, and scholarship award of the Interdisciplinary Graduate School of Energy System, Prince of Songkla University.

**References**
[1] Mujumdar, A.S. (2006). *Principles, classification, and selection of dryers*. Handbook of industrial drying, Page: 4-31.
[2] Jeantet, R. Croguennec, T. and Schuck, P. (2016). *Food Process Engineering and Packaging: John Wiley & Sons.* Handbook of Food Science and Technology 2, Page: 63.
[3] Keey, R.B. (2013). *Drying: principles and practice*: Elsevier, Page: 178.
[4] Liptak, B.G. (1998). *Optimization of industrial unit processes*: CRC press, Page: 1161-1162.
[5] Scherer, V. Monnigmann, M. Berner, M.O. and Sudbrock F. "Coupled DEM–CFD simulation of drying wood chips in a rotary drum–baffle design and model reduction." *Fuel 184* (2016): 896-904.
[6] Machado, M. Nascimento, S. Duarte, C. and Barrozo, M. "Boundary conditions effects on the particle dynamic flow in a rotary drum with a single flight." *Powder Technology* 311 (2017): 341-9.
[7] Ajayi, O. and Sheehan, M. "Design loading of free flowing and cohesive solids in flighted rotary dryers." *Chemical engineering science* 73 (2012): 400-11.
[8] O'Connor, D. Calautit, J.K. and Hughes, B.R. "A novel design of a desiccant rotary wheel for passive ventilation applications." *Applied Energy* 179 (2016): 99-109.
[9] Alonso, E. Gallo, A. Roldán, M. Perez-Rabago, C. and Fuentealba, E. "Use of rotary kilns for solar thermal applications: Review of developed studies and analysis of their potential." *Solar Energy* 144 (2017): 90-104.
[10] Delele, M.A, Weigler, F. Franke, G. and Mellmann, J. "Studying the solids and fluid flow behavior in rotary drums based on a multiphase CFD model." *Powder Technology* 292 (2016): 260-71.
[11] Santos, D. Duarte, C. and Barrozo, M. "Segregation phenomenon in a rotary drum: Experimental study and CFD simulation." *Powder Technology* 294 (2016): 1-10.
[12] Machado, M. Nascimento, S. Duarte, C. and Barrozo, M. "Boundary conditions effects on the particle dynamic flow in a rotary drum with a single flight." *Powder Technology* 311 (2017): 341-9.
[13] Liu, T. Qi, X. Su, G. Jin, J. and Cong, W. "Computational fluid dynamics (CFD) simulation of flow in the rotary drum for pyrite bio-preoxidization." *Chemical Engineering and Processing: Process Intensification* 47(5) (2008): 971-8.
[14] Geng, F. Chai, H. Ma, L. Luo, G. Li, Y. and Yuan, Z. "Simulation of dynamic transport of flexible ribbon particles in a rotary dryer." *Powder Technology* 297 (2016): 115-25.
[15] Geng, F. Yuan, Z. Yan, Y. Luo, D. Wang, H. Li, B. and Xu, D.Y. "Numerical simulation on mixing kinetics of slender particles in a rotary dryer." *Powder Technology* 193(1) (2009): 50-8.
[16] Gu, C. Li, P. Yuan, Z. Yan, Y. Luo, D. Li, B. and Lu, D. "A new corrected formula to predict mean residence time of flexible filamentous particles in rotary dryers." *Powder Technology* 303 (2016): 168-75.
[17] Gu, C. Zhang, X. Li, B. and Yuan, Z. "Study on heat and mass transfer of flexible filamentous particles in a rotary dryer." Powder *Technology* 267 (2014): 234-9.
[18] Geng, F. Li, Y. Wang, X. Yuan, Z. Yan, Y. and Luo, D. "Simulation of dynamic processes on flexible filamentous particles in the transverse section of a rotary dryer and its comparison with ideo-imaging experiments." *Powder technology* 207(1) (2011): 175-82.
[19] Karunarathne, S.S. Jayarathna, C.K. and Tokheim, L.A. "Mixing and Segregation in a Rotating Cylinder: CFD Simulation and Experimental Study." *International Journal of Modeling and Optimization* 7(1) (2017): 1.
[20] Nafsun, AI. Herz, F. and Liu, X. "Influence of material thermal properties and dispersity on thermal bed mixing in rotary drums." *Powder Technology* 331 (2018): 121-128.
[21] Hanifarianty, S. Legwiriyakul, A. Alimalbari, A. Nuntadusit, C. Theppaya, T. and Wae-Hayee, M. "The development of rotary drum dryer for palm fruit sterilization." *IOP Conference Series: Materials Science and Engineering: IOP Publishing* (2018), Page: 012031.

[22] Hanifarianty, S. Legwiriyakul, A. Alimalbari, A. Nuntadusit, C. Theppaya, T. and Wae-Hayee, M. “A rotary drum dryer for palm sterilization: preliminary study of flow and heat transfer using CFD.” *IOP Conference Series: Materials Science and Engineering: IOP Publishing* (2018), Page: 012030.

[23] Hassan, IA. A, Sadikin. and N, Mat Isac. "The Computational Modeling of Falling Film Thickness Flowing Over Evaporator Tubes." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences,* 14(1) (2015): 24-37.

[24] Pairan. Rasidi, Mohamad. Asmuinb, N. and Salleh, Hamidon. "Simulation of Film Cooling in the Leading Edge Region of a Turbine Blade Using ANSYS CFX." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* (1) (2014): 1-10.

[25] Sapee, S. "Computational Fluid Dynamics Study on Droplet Size of Kerosene Fuel." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* (16) (2015): 1-14.

[26] Too. JHY. and CSN, Azwadi. "Numerical Analysis for Optimizing Solar Updraft Tower Design Using Computational Fluid Dynamics (CFD)." *Journal Advance Research Fluid Mechanics and Thermal Science* 22 (2016): 8-36.

[27] Zakaria. Shukri, Mohamad. Ismail, Farzad. Tamagawa, Masaaki. Azi, Ahmad Fazli Abdul. Wiriadidjaya, Surjatin. Basri, Adi Azrif. and Ahmad, Kamarul Arifin." Computational Fluid Dynamics Study of Blood Flow in Aorta using OpenFOAM." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 43 (1) (2018): 81-89.

[28] Cengel, Y. (2015). *Heat and mass transfer: fundamentals and applications*. Fifth Edition in SI Units: McGraw-Hill Higher Education, Page: 531 – 541.