COMPUTATIONAL FLUID DYNAMICS ANALYSIS FOR IMPROVING TEMPERATURE DISTRIBUTION IN A CHILI DRYER

by

Jose L. CARRERA ESCOBEDO[a*], Arquimedes ORTIZ RIVERA[a], Cesar H. GUZMAN VALDIVIA[b], Mario A. GARCIA RUIZ[a], and Omar DESIGA ORENDAY[c]

[a] Department of Mechatronics Engineering, Polytechnic University of Zacatecas, Zacatecas, Mexico
[b] Faculty of Electrical Engineering, Autonomous University of Zacatecas, Zacatecas, Mexico
[c] Academy of Mechatronics Engineering, Interdisciplinary Professional Unit on Engineering, Campus Zacatecas, Zacatecas, Mexico

Original scientific paper
https://doi.org/10.2298/TSCI160112255E

Computational fluid dynamics is a numerical tool that is highly accurate to simulate a very large number of applications and processes. The CFD analysis has emerged as a viable technique to provide effective and efficient design solutions. In this paper, a CFD analysis for improving temperature distribution in a chili dryer is presented. The CFD technique is used to simulate the temperature distribution inside the chamber. For this purpose, the continuity, momentum and energy equations are considered. The results obtained by CFD analysis based on a specific geometry are presented in order to improve the temperature distribution. In addition, these results were verified experimentally. The distribution of temperatures showed small differences around 4 K during the warming up period. The simulation and experimental results can be useful for further designs of chili dryers with different specific geometries.

Key words: *airflow, computational fluid dynamics, chili dryer, design, k-ε, simulation, temperature distribution*

Introduction

Chili (*Capsicum annuum L.*) is one of the most important vegetables in Mexico with a consumption of 0.56 kg per capita [1]. It is consumed both fresh or as dried product [2]. Chili drying is often used for preservation and allows reducing production costs [3]. However, this traditional process requires enormous amounts of energy for heating. The chili drying process requires fossil fuels which contributes to global warming. Therefore, the improvement of sustainable energy systems such as the chili dryers are a very interesting engineering field due to their benefits to the environment. In order to reduce the energy consumption, it is necessary to design a dryer with an efficient temperature distribution inside the chamber. One of the main difficulties in the design of these dryers is to keep a uniform and stable distribution of temperatures [4, 5].

In recent years, considerable efforts have been made to design and develop many chili dryers. These dryers can be classified in natural and forced convection [6, 7]. In [8, 9] some

* Corresponding author, e-mail: jlcarrera20@hotmail.com

natural convection systems were proposed. On the other hand, the forced convection dryers are generally more effective and controllable to provide the optimum temperature. Mohanraj and Chandrasekar [10] developed an indirect forced convection solar dryer integrated with different sensible heat storage material for drying chili. Charmongkolpradit *et al.* [11] and Cortes *et al.* [12] investigated the drying characteristics of chili using a continuous fluidized-bed dryer. Marnoto *et al.* [13] investigated the advantages of a dryer using a heat pump dehumidifier. Umayal *et al.* [14] designed and fabricated a solar dryer with an evacuated tube collector and a heat storage material. Hudakorn and Katejanekarn [15] studied the thermal performance of a square-corrugated air collector with attached internal fins for red chili drying. Banout *et al.* [16] proposed a double-pass solar dryer and Kaensup *et al.* [17] designed a microwave-vacuum system for drying red chili. Kaleemullah and Kailappan [18] investigated the drying kinetics of red chilies using a rotary dryer. Kaewkiew *et al.* [19] investigated the performance of a large-scale solar dryer greenhouse for chili drying. In [20, 21] some solar tunnel dryers for chili drying were studied. Other simple forced convective solar chili dryers were proposed in [22, 23].

In most forced convection chili dryers, hot air flow is usually introduced through the chamber and passes over the tray. However, the temperature distribution may not be uniform due to the location of the chilies. Therefore, the chilies at the beginning of the dryer may be over dried, while the chilies located on the end of the dryer may not receive enough heat. In this kind of dryers, the drying air temperature shows a very strong relationship with the dehydration rate [24, 25]. The uniform temperature and air flow distribution determines both the efficiency and the uniformity of the products [26]. However, all these parameters are very difficult to find experimentally. On the other hand, the CFD is a non expensive and low time-consuming tool that uses numerical analysis and algorithms to solve problems that involve fluid flows. The CDF does the calculations required to simulate the flow patterns with surfaces defined by boundary conditions, which was previously impossible from experimental and theoretical methods [27, 28]. With the increasing availability of power computers, there has been a considerable growth in the application of CFD analysis [29]. As a result, it predicts the performance of new process designs before manufacture or implementation [30].

Since the CFD analysis has been used for predicting the air velocity and temperature distribution in a large number of applications, many researchers have shown interest in this topic. Thakker and Elhemry [31] proposed a CFD analysis based on an impulse turbine with different hub-to-tip ratio for wave energy conversion. Westerlund *et al.* [32] investigated the optimization of a pellet burner using a simplified CFD model. Gallegos *et al.* [33] analyzed the effect caused by fitting in the measurements of flow in an air conditioning system. Okita *et al.* [34] investigated the heat transfer process during the freezing of guava pulp contained in a stack of large containers. Recently, several studies have reported the analysis of different kinds of dryers for agricultural products using CFD. Amanlou and Zomorodian [35] proposed a fruit cabinet dryer with a side mounted plenum chamber. Margaris and Ghiaus [36] investigated the dried product quality improvement by air flow manipulation in tray dryers. Mathioulakis *et al.* [37] investigated and simulated the air movement inside an industrial batch-type tray dryer. Prukwarun *et al.* [38] reported a CFD simulation of fixed-bed dryer by using porous media concepts. Roman *et al.* [39] studied the improvement of air distribution in a fixed-bed dryer using CFD. Mirade and Daudin [40] presented a study of the airflow patterns in a sausage dryer by comparing numerical results with experimental air speed data. Weigler *et al.* [41] proposed experimental studies on a mixed-flow dryer. Tzempelikos *et al.* [42] presented an analysis of air velocity distribution in a laboratory batch-type-tray dryer. Other similar types of dryers were proposed in [43, 44]. In these studies, CFD has proven to be an important simulation tool for the analysis and improvement of such dryers.

Based on the review of the state-of-the-art, it was found that there is a scope for enhancing the temperature distribution in a chili dryer. The main objective of this study is to improve the temperature distribution in a chili dryer considering the effects of continuous air speed to guarantee uniformity inside the chamber with a minimum energy consumption using CFD analysis.

Methodology

Governing equations

Heat and mass transfer are accomplished by solving the continuity, momentum and energy equations with boundary conditions. In this study, the fluid was considered incompressible. These equations are expressed:

– continuity equation

$$\nabla(\rho\vec{\mathrm{u}}) = 0 \tag{1}$$

– momentum equation

$$\nabla(\rho\vec{\mathrm{u}}\vec{\mathrm{u}}) = -\nabla p + \nabla(\boldsymbol{\tau}) + \rho\vec{\mathrm{g}} \tag{2}$$

– energy equation

$$\begin{gathered}\nabla(\rho\vec{\mathrm{u}}H) = \nabla\left(\frac{k_t}{C_p}\nabla H\right) + S_h \\ H = \int_{T_0}^{T} C_p \mathrm{d}T\end{gathered} \tag{3}$$

where $\vec{\mathrm{u}}$ is the velocity vector, ρ – the density, p – the pressure, $\vec{\mathrm{g}}$ – the gravity, μ – the viscosity, H – the enthalpy, k_t – the thermal conductivity, C_p – the specific heat, S_h – a source term, T – the temperature, I – the identity matrix, and $\boldsymbol{\tau}$ – the stress tensor defined as:

$$\boldsymbol{\tau} = \mu\left\{\left[(\nabla\vec{\mathrm{u}}) + (\nabla\vec{\mathrm{u}})^T\right] - \frac{2}{3}\nabla\vec{\mathrm{u}}\mathrm{I}\right\} \tag{4}$$

The air flow proposed is turbulent. Therefore, the standard κ-ε model was used because it is the most widely validated turbulence model in literature [45-47]. In the κ-ε model, κ is the turbulence kinetic energy and ε is the turbulent dissipation.

Numerical methods and boundary conditions

To assess the temperature distribution inside the chili dryer with an specific geometry, the governing equations were solved numerically using the commercial CFD code ANSYS-FLUENT® (version 6.3). This CFD code uses the finite volume method (FVM). The FVM uses a volume integral formulation of the problem with a finite partitioning set of volumes to discretize the partial differential equations. The QUICK (quadratic upstream interpolation for convective kinematics) scheme was used for the cell phase values and the semi-implicit method for pressure-linked equations (SIMPLE) algorithm was used to solve the pressure-velocity coupling equations [48]. The first proposed design is shown in fig. 1. The dimensions of the chamber were 0.5 m (length) × 0.4 (width) × 0.3 m (height) to han-

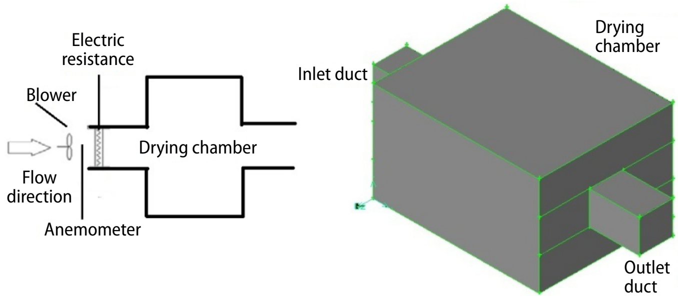

Figure 1. First proposed design; (a) overview of the components, (b) computational model

dle chili samples up to 2 kg of weight. The dimensions of the rectangular cross-section of inlet and outlet ducts were 0.1 m × 0.1 m.

The CFD analysis require boundary conditions, in particular with the surface bounding of the domain. For this study, the boundary conditions were defined as follows.

– *Walls*. A no slip shear condition was selected. The wall temperatures were set at 293 K (room temperature). The entire chamber was thermally insulated with a layer of glass wool (0.0254 m of thickness) with a thermal conductivity of 0.03 W/mK. The thermal resistance was set at 0.847 K/W and the heat-transfer coefficient was set at 1.18 W/m^{2}K.
– *Inlet*. The inlet temperature was set at 333 K. The hydraulic diameter of the inlet duct was 0.1 m. The turbulence intensity was set at 5%.
– *Outlet*. If assuming gauge pressure = 0 , then the required information will be extrapolated from the interior of the drying chamber using the CFD software.

Results

Initial CFD analysis

The CFD was used for the analysis of temperature distribution inside the chamber. A mesh sensitivity analysis was done to find the size and number of cells suitable for this analysis. The relation between size and number of cells is shown in tab. 1. For the simulations, a desktop PC (Intel® Core ™ i5 CPU at 2.67 GHz with 8 GB of RAM) was used. The proposed inlet air speed for this simulation was set at 0.1 m/s. The first design was used. Figure 2 shows the temperature contours using different mesh sizes. It is observed that the temperature distribution is more uniform using 63000 and 87274 cells and the higher values are located along the x-axis. As it can be seen in fig. 2, the x-axis is always parallel to the flow direction. Thus, the higher temperature zone is located at the center of the chili dryer. To determine the acceptable mesh size for the numerical study a second analysis is needed.

Table 1. Relation of size and number of cells

Size of cells	0.03 m	0.02 m	0.01 m	0.009 m
Number of cells	2079	8275	63000	87274

Figure 3 shows the temperature distribution using different mesh sizes. Comparing both figs. 2 and 3, there is not a significant difference using more than 63000 cells. For this reason, this mesh was used for the next CFD analysis. On the other hand, the temperature distribution obtained was not satisfactory due to the low uniformity inside the chamber. The difference of temperatures was 13 K. Considering these results, it was necessary to modify physical parameters of the chili dryer to improve the temperature distribution.

Improving the chili dryer

The outlet duct size

Considering the previous CFD simulation, the first improvement for the chili dryer is to modify the outlet duct area. The new dimensions for the proposed rectangular cross-section

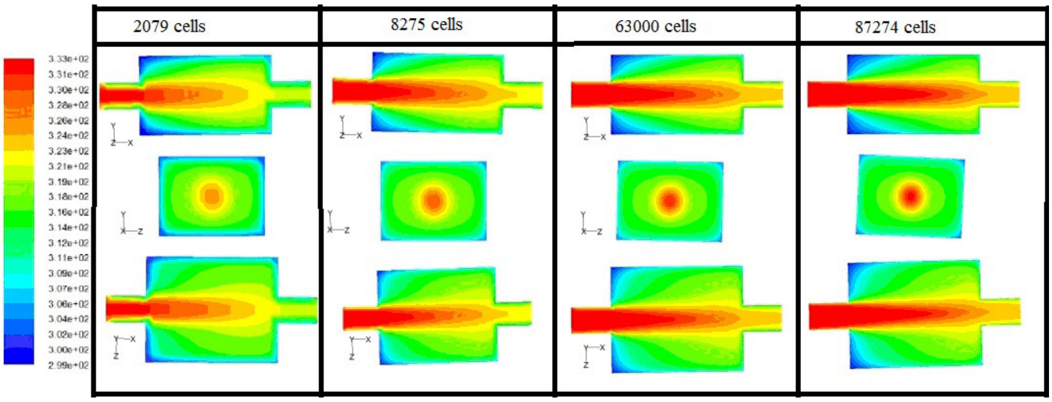

Figure 2. Temperature contours using different mesh sizes
(*for color image see journal web site*)

outlet ducts are 0.0025 m^2 and 0.0004 m^2. The same inlet air speed (0.1 m/s) was used in this simulation. Figure 4 shows the temperature distribution using different outlet duct areas. It is observed that the best temperature distribution was reached using the minimum area. However, the distribution is not uniform and it is necessary to add a diffuser.

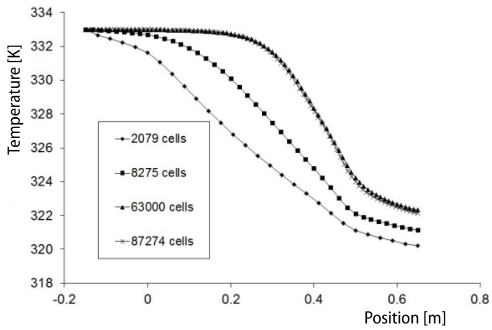

Figure 3. Temperature distribution using different mesh sizes

Adding a diffuser

Three different angles of cone (α = 90°, α = 60°, and α = 40°) were proposed. The sketch of the diffuser angle is shown in fig. 5. The same inlet air speed (0.1 m/s) was used in this simulation. Figure 6 shows the temperature contours using a diffuser. It is observed that using an angle α = 90° the dissemination point is reduced. On the other hand, using an angle α = 40° the temperature distribution is more uni-

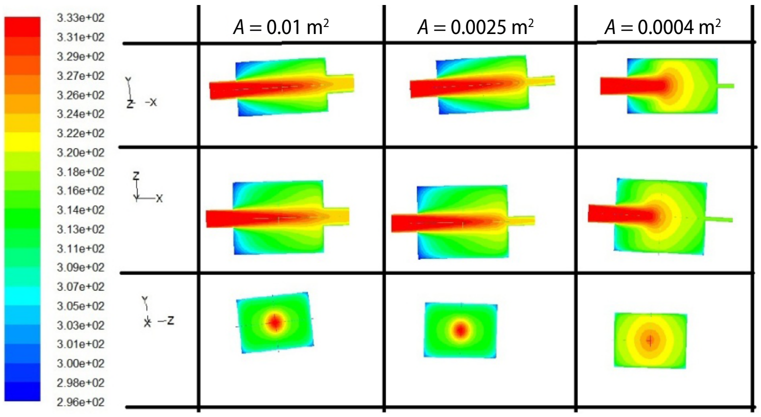

Figure 4. Temperature contours using different outlet duct areas
(*for color image see journal web site*)

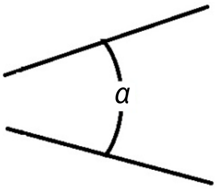

Figure 5. Sketch of the diffuser angle

form. However, the temperature distribution has a difference of 17 K and does not cover the entire chamber. For this reason, it is necessary to do another simulation considering the inlet air speed.

Inlet air velocity

Three different inlet air speeds (0.2 m/s, 0.4 m/s, and 0.5 m/s) were proposed. Figure 7 shows the temperature contours using different inlet air speeds. It is observed that using the higher inlet air sèed the temperature distribution becomes more uniform inside the chili dryer. To determine the acceptable temperature distribution for the chili dryer a second analysis is needed.

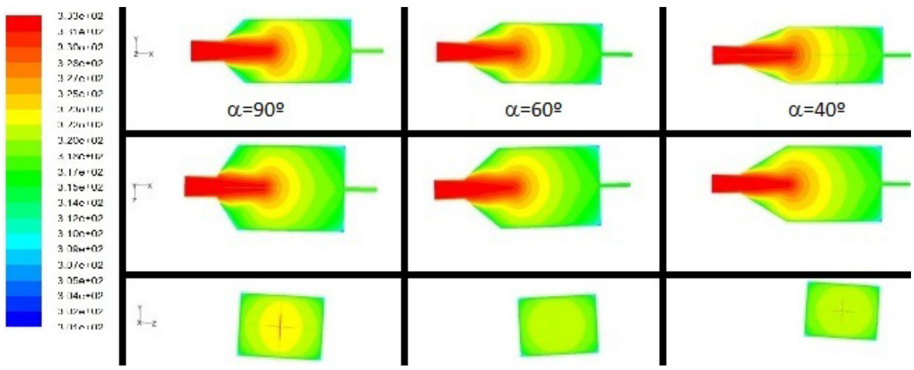

Figure 6. Temperature contours using a diffuser
(*for color image see journal web site*)

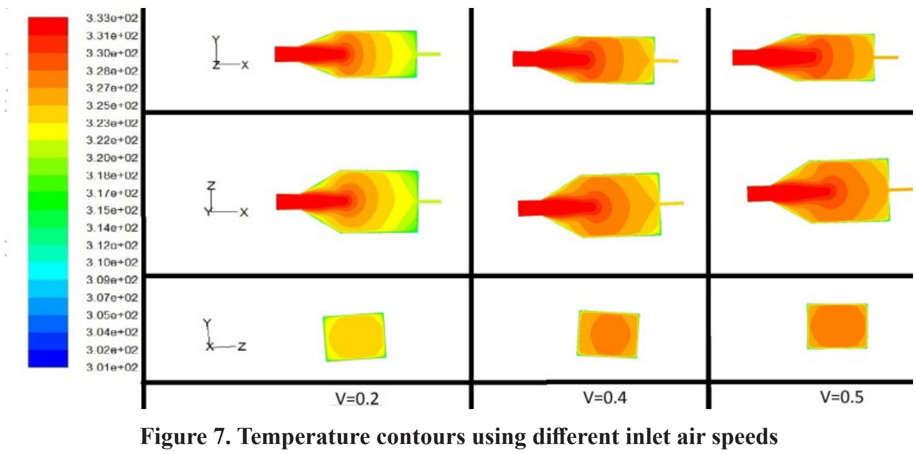

Figure 7. Temperature contours using different inlet air speeds
(*for color image see journal web site*)

Figure 8 shows the temperature distribution at the center line of the dryer, using different inlet air speeds. Comparing both figs. 7 and 8, there is a small difference of 6 K with an inlet air speeds of 0.5 m/s. The temperature distribution obtained was satisfactory due to the high uniformity inside the chamber. Considering these results, it is necessary to do an experimental validation.

Experimental validation of temperature distribution

The chili dyer was manufactured considering the improvements for the outlet duct size and the addition of a diffuser. To measure the temperature inside the chamber, six analog thermocouples type K were installed as shown in fig. 9. Table 2 shows the technical information of the temperature sensors.

Figure 10 shows the temperature distribution during the warming up period. It is observed that the distribution of temperatures shows small differences around 4 K during the warming up period. These experimental results validate the simulation results.

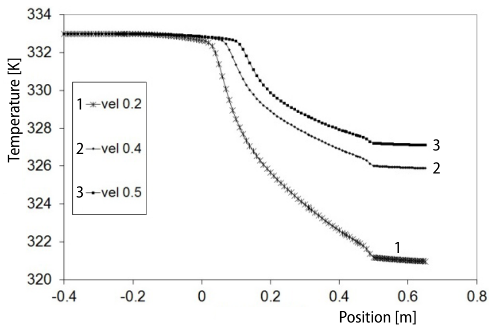

Figure 8. Temperature distribution using different inlet air speeds

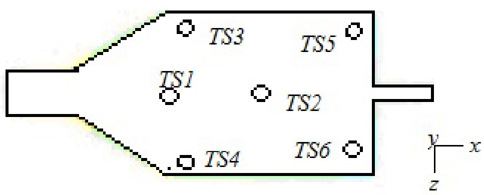

Figure 9. Location of the temperature sensors

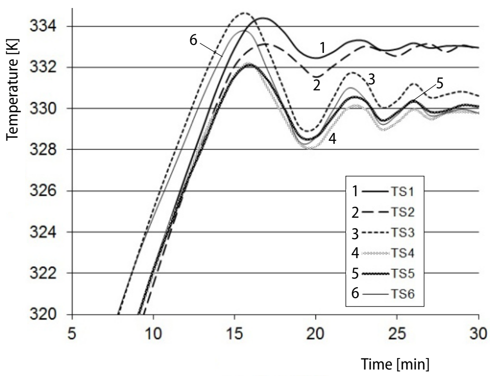

Figure 10. Temperature distribution during the warming up period

Table 2. Model and parameter information of the temperature sensors

Model	Resolution	Operating range	Accuracy
Thermocouple type K	24-bit	–270 °C to 1372 °C	±0.345 °C

Conclusions

In this paper, a CFD analysis for improving temperature distribution in a chili dryer was presented. The continuity, momentum, and energy equations were solved using the CFD software. The first proposed design was modified to improve the temperature distribution. The improvements for the chili dryer were done in the outlet duct size and the addition of a diffuser. Additionally, the effects of different inlet air speeds were considered to improve the temperature uniformity inside the chamber. The experimental validation of the temperature distribution was done and compared with the simulation results extracted from the CFD analysis. This comparison was satisfactory and acceptable due to the small temperature differences. The next study will undertake the addition of a recirculation system. These findings will be useful for further designs of chili dryers with different specific geometries.

Acknowledgment

The authors thank Editor-in-chief and anonymous reviewers for their constructive comments. The financial support from the National Council of Science and Technology (CONACYT) with project number 200679 is gratefully acknowledged. Finally, the authors thank Vanesa Robles Maldonado for her support in improving the writing of this paper.

Nomenclature

C_p – specific heat, [kJ kg^{-1} K^{-1}]
$\vec{g}$ – gravity, [ms^{-2}]
H – enthalpy, [kJkg^{-1}]
k_t – thermal conductivity, [Wm^{-1}K^{-1}]
p – pressure, [Nm^{-2}]
T – temperature, [K]
S_h – energy source term, [wm^{-3}]
$\vec{u}$ – velocity vector, [ms^{-1}]

Greek symbols

ρ – density, [kgm^{-3}]
$\boldsymbol{\tau}$ – stress tensor, [Nm^{-2}]

References

[1] Gonzalez, G., The Technical Assistance Service to the Producers of Dry Chili in Zacatecas, *Convergence, 14* (2007), 43, pp. 137-165
[2] Gaytan, D., Benita, F., On the Competitiveness of Mexico's Dry Chili Production, *Economics of Agriculture*, *61* (2014), 2, pp. 307-317
[3] Hossain, M., Bala, B., Thin-Layer Drying Characteristics for Green Chilli, *Drying Technology*, *20* (2002), 2, pp. 489-505
[4] Arora, S., *et al.*, Convective Drying Kinetics of Red Chilies, *Drying Technology*, *24* (2006), 2, pp. 189-193
[5] Gupta, P., *et al.*, Drying Characteristics of Red Chilli, *Drying Technology*, *20* (2002), 10, pp. 1975-1987
[6] VijayaVenkataRaman, S., *et al.*, A Review of Solar Drying Technologies, *Renewable and Sustainable Energy Reviews*, *16* (2012), 5, pp. 2652-2670
[7] El, A., Shalaby, S., Solar Drying of Agricultural Products: A Review, *Renewable and Sustainable Energy Reviews*, *16* (2012), 1, pp. 37-43
[8] Hossain, M., *et al.*, Design and Development of Solar Dryer for Chilli Drying, *International Journal of Research*, *2* (2015), 1, pp. 63-78
[9] Papade, C., Boda, M., Design and Development of Indirect Type Solar Dryer with Energy Storing Material, *International Journal of Innovative Research in Advanced Engineering*, *1* (2014), 12, pp. 109-114
[10] Mohanraj, M., Chandrasekar, P., Performance of a Forced Convection Solar Drier Integrated with Gravel as Heat Storage Material, *Journal of Engineering Science and Technology*, *4* (2009), 3, pp. 305-314
[11] Charmongkolpradit, S., *et al.*, Drying Characteristics of Chili Using Continuous Fluidized-Bed Dryer, *American Journal of Applied Sciences*, *7* (2010), 10, pp. 1300-1304
[12] Cortes, E., *et al.*, Feasibility Analysis of Drying Process Habanero Chili Using a Hybrid-Solar-Fluidized Bed Dryer in Yucatan, Maxico, *Journal of Energy and Power Engineering*, *7* (2013), 10, pp. 1898-908
[13] Marnoto, T., *et al.*, The Characteristic of Heat Pump Dehumidifier Drier in the Drying of Red Chili (*Capsicum Annum L*), *International Journal of Science and Engineering*, *3* (2012), 1, pp. 22-25
[14] Umayal, A., *et al.*, Performance of Evacuated Tube Collector Solar Dryer with and without Heat Sources, *Iranica Journal of Energy & Environment*, *4* (2013), 4, pp. 336-342
[15] Hudakorn, T., Katejanekarn, T., Performance of a Square-Corrugated Air Collector with Attached Internal Fins Solar Drier for Red Chili Drying, *Journal of Science and Technology*, *31* (2012), 5, pp. 592-597
[16] Banout, J., *et al.*, Design and Performance Evaluation of a Double-Pass Solar Drier for Drying of Red Chilli (Capsicum Annum L), *Solar Energy*, *85* (2011), 3, pp. 506-515
[17] Kaensup, W., *et al.*, Experimental Study on Drying of Chilli in a Combined Microwave-Vacuum-Rotary Drum Dryer, *Drying Technology*, *20* (2002), 10, pp. 2067-2079
[18] Kaleemullah, S., Kailappan, R., Drying Kinetics of Red Chillies in a Rotary Dryer, *Biosystems Engineering*, *92* (2005), 1, pp. 15-23
[19] Kaewkiew, J., *et al.*, Experimental Investigation of the Performance of a Large-Scale Greenhouse Type Solar Dryer for Drying Chilli in Thailand, *Procedia Engineering*, *32* (2012), 1, pp. 433-439
[20] Hossain, M., Bala, B., Drying of Hot Chilli Using Solar Tunnel Drier, *Solar Energy*, *81* (2007), 1, pp. 85-92
[21] Hossain, M., *et al.*, Simulation of Solar Drying of Chilli in Solar Tunnel Drier, *Drying Technology*, *24* (2005), 3, pp. 143-153

[22] Keawsuntia, Y., Experimental Investigation of Active Solar Dryer for Drying of Chili, *Advanced Materials Research*, *953* (2014), 1, pp. 16-19
[23] Fudholia, A., *et al*., Performance Analysis of Solar Drying System for Red Chili, *Solar Energy*, *99* (2014), 1, pp. 47-54
[24] Karathanos, V., Belessiotis, V., Sun and Artificial Air Drying Kinetics of Some Agricultural Products, *Journal of Food Engineering*, *31* (1997), 1, pp. 35-46
[25] Mulet, A., *et al*., Effect of Air Flow Rate on Carrot Drying, *Drying Technology*, *5* (1987), 2, pp. 245-258
[26] Mirade, P., Prediction of the Air Velocity Field in Modern Meat Dryers Using Unsteady Computational Fluid Dynamics (CFD) Models, *Journal of Food Engineering*, *60* (2003), 1, pp. 41-48
[27] Anderson, J., *Computational Fluid Dynamics*, McGraw-Hill, New York, USA, 1995
[28] Yongson, O., *et al*., Airflow Analysis in an Air Conditioning Room, *Building and Environment*, *42* (2007), 3, 1531-1537
[29] Parviz, M., John, K., Tackling Turbulence with Supercomputers, *Scientific American*, *276* (1997), 1, pp. 62-68
[30] Schaldach, G., *et al*., *Computer Simulation for Fundamental Studies and Optimization of ICP Spray Chambers*, Institute of Spectrochemistry and Applied Spectroscopy, Current Research Reports, Dortmund, Germany, 2000
[31] Thakker, A., Elhemry, M., 3-D CFD Analysis on Effect of Hub-to-Tip Ratio on Performance of Impulse Turbine Wave Energy Conversion, *Thermal Science, 11* (2007), 4, pp. 157-170
[32] Westerlund, L., *et al*., Computational Fluid Dynamics Optimization of a Pellet Burner, *Thermal Science, 16* (2012), 4, pp. 1175-1186
[33] Gallegos, A., *et al*., Analysis of Effect Caused by Fitting in the Measurements of Flow in Air Conditioning System, *Applied Thermal Engineering*, *33* (2012), 1, pp. 227-236
[34] Okita, W., *et al*., Heat Transfer Analyses Using Computational Fluid Dynamics in the Air Blast Freezing of Guava Pulp in Large Containers, *Brazilian Journal of Chemical Engineering*, *30* (2013), 4, pp. 813-824
[35] Amanlou, Y., Zomorodian, A., Applying CFD for Designing a New Fruit Cabinet Dryer, *Journal of Food Engineering*, *101* (2010), 1, pp. 8-15
[36] Margaris, D., Ghiaus, A., Dried Product Quality Improvement by Air Flow Manipulation in Tray Dryers, *Journal of Food Engineering, 75* (2006), 4, pp. 542-550
[37] Mathioulakis, E., *et al*., Simulation of Air Movement in a Dryer by Computational Fluid Dynamics: Application for the Drying of Fruits, *Journal of Food Engineering, 36* (1998), 2, pp. 183-200
[38] Prukwarun, W., *et al*., CFD Simulation of Fixed Bed Dryer by Using Porous Media Concepts: Unpeeled Longan Case, *International Journal of Agricultural and Biological Engineering, 6* (2013), 1, pp. 100-110
[39] Roman, F., *et al.,* Improvement of Air Distribution in a Fixed-Bed Dryer Using Computational Fluid Dynamics, *Biosystems engineering*, *122* (2012), 4, pp. 359-369
[40] Mirade, P., Daudin, J., A Numerical Study of the Airflow Patterns in a Sausage Dryer, *Drying Technology, 18* (2000), 1, pp. 81-97
[41] Weigler, F., *et al.,* Experimental Studies on a Newly Developed Mixed-Flow Dryer, *Drying Technology, 31* (2013), 15, pp. 1736-1743
[42] Tzempelikos, D., *et al.,* Analysis of Air Velocity Distribution in a Laboratory Batch-Type Tray Air Dryer by Computational Fluid Dynamics, *International Journal of Mathematics and Computer in Simulation*, *5* (2012), 6, pp. 413-421
[43] Aissa, W., *et al.,* Performance of Solar Dryer Chamber Used for Convective Drying of Sponge-Cotton, *Thermal Science*, *18* (2014), 2, pp. 451-462
[44] Zdanski P., *et al.,* Numerical Assessment of the Air Flow Behavior in a Conventional Compact Dry Kiln., *Journal of Applied Fluid Mechanics*, *8* (2015), 3, pp. 367-376
[45] Versteeg, H., Malalasekera W., *An Introduction to Computational Fluid Dynamics the Finite Volume Method*, Pearson Education Limited., London, 2007
[46] Launder, E., Spalding B., *Lectures in Mathematical Models of Turbulence*, Academic Press, London, 1972
[47] Cengel, Y., Cimbala, J., *Fluid Mechanics Fundamental an Applications*, McGraw-Hill, New York, USA, 2015
[48] Patankar, S., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington D. C., USA, 1980

Paper submitted: January 12, 2016
Paper revised: September 25, 2016
Paper accepted: October 14, 2016