



Available online at www.sciencedirect.com



Procedia

Energy Procedia 110 (2017) 465 - 470

1st International Conference on Energy and Power, ICEP2016, 14-16 December 2016, RMIT University, Melbourne, Australia

Preliminary investigation of the flow distribution in an innovative intermittent convective microwave dryer (IMCD)

Zachary Welsh, Chandan Kumar, Azharul Karim*

Science & Engineering Faculty, Queensland University of Technology, 2 George St, Brisbane 4000, Australia

Abstract

Intermittent Microwave Convection Drying (IMCD) is an innovative concept that has the potential to reduce drying time and improve both energy efficiency and product quality. However very few studies on flow or temperature distributions on IMCD can be found in the literature. In this study, an innovative U shaped continuous flow IMCD has been proposed. In order to take full advantage of the IMCD technology, a uniform airflow distribution in the drying chamber is critical. Uneven airflow in the drying chamber significantly deteriorate the food quality and energy efficiency and increase the drying time. This research investigates the flow distribution in the proposed U shaped IMCD using computational fluid dynamics (CFD). Effects of varying inlet air velocity on the airflow distribution were investigated. The geometry, mesh and turbulence model were kept constant throughout the investigation. COMSOL Multiphysics software alongside a SST k- ω turbulence model was utilised. The investigation revealed that the distribution of the U shaped geometry minimises the 'dead flow' regions compared to rectangular shaped geometry. However, two small 'dead zones' were still found in U shaped geometry. This study further investigates how these dead zones can be minimised by changing the location of inlets and outlets.

© 2017 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Peer-review under responsibility of the organizing committee of the 1st International Conference on Energy and Power.

Keywords: Flow distribution, IMCD, CFD

1. Introduction

Dried fruit and vegetable industry has great potential in Australia, with the demand for high quality and high value food constantly growing. This sector for Australia has a gross value of \$A9 billion, with Queensland having the largest

^{*} Corresponding author. Tel.: +61 7 3138 1516. *E-mail address:* azharul.karim@qut.edu.au

contribution of the industry of 32% [1]. Traditional convection drying techniques are currently the most dominant process in the industry however, these techniques are time consuming and energy intensive and can often result in 40% of the produce becoming waste during the food cycle [2,3,4,5,6,7,8]. If Australia is to reach its full potential in this industry, the development of new processes that sustainably deliver high quality and high value food will be essential.

Intermittent Microwave Convection Drying (IMCD) is an innovative concept that has the potential to reduce drying time and improve both energy efficiency and product quality. Computational Fluid Dynamics (CFD) has become a common resource to predict/analysis air velocity and temperature distribution within drying chambers [9]. One style of dryer, which is quite common but generally designed without any investigations into the airflow distribution, is tray dryers. Tray dryers are typically used in the industry due to three main features: simple design, high volume and lost cost [10]. There are two main variables that will have a major effect upon the airflow distribution of a tray style dryer; the geometry and arrangement.

The geometry has a major effect on the airflow distribution within a dryer. Various geometries, from cylinders [11] to rectangular tray dryers [12], have been investigated and their effects have been reported in the literature. Amanlou, and Zomorodian [13] investigated different designs of dryers to achieve uniform moisture content within the end product. Simulations on several different geometries were conducted and their associated affects upon the resulting airflow distribution were discussed in detail. The different geometries simulated are all slight modifications to the main geometry which included altering the design of the corners, the angle of the plenum chamber. The arrangement of the inlet/outlet locations within a dryer can also have a great effect upon the airflow distribution. The effect inlet/outlet arrangement will depend on a few key variables; location/position [11,13,14], size/number [15] and direction [16]. However, all these studied were conducted on traditional convective dryer. No study has investigated flow, temperature and moisture distribution in an IMCD, particularly in U shaped continuous IMCD.

This paper aims to investigate the air flow distribution of a U shaped continuous IMCD dryer using computational fluid dynamics (CFD). The flow distribution will be investigated in COMSOL Multiphysics software. To investigate the flow distribution the air's inlet velocity will be varied along with two separate inlet/outlet configuration will be simulated. This will allow the flow distribution to be investigated in great depth. Three velocities will be investigated: 0.5 m/s, 2.5 m/s and 5 m/s.

Nomenclature						
Re	Reynold Number					
ρ	Density of Air					
u	Dynamic Viscosity of Air					
v	Velocity					
D_h	Hydraulic Diameter					
Р	Pressure					

2. Geometry

As mentioned earlier, a U shaped tray style continuous dryer has been investigated. At this stage of the preliminary investigation the dryer was investigated and modelled as an empty dryer. The trays and tray holders will have a significant effect on the air flow distribution and will require further in-depth investigation, which will be done later stage of this research. The two different inlet/outlet locations, as can be seen in Figure 1, were studied. In the figure, the orange colour represents the inlets locations and yellow represents the outlets locations. Configuration 1 has a single inlet and outlet located at opposite ends of the U. Configuration 2 has two inlets (one at each end of the U) and a single outlet located at the top centre of the bend. The dryer is a square channel of 500 mm by 500 mm with an overall size of 2590 mm x 2180mm (L x W). For the CFD simulations the domain of the geometry was kept only to the inside area of the dryer.



Fig 1. Inlet/Outlet locations - Orange = inlet, Yellow = Outlet (a) Configuration 1 (b) Configuration 2

3. Model Development

Before the model development process could begin, the conditions of the flow field within the dryer had to be classified. The Reynold number was used to classify the flow and can be seen in equation (1). The Reynold numbers for all six simulations are listed in Table 1. The properties of air at 50°C were used in the calculations [17].

$$\operatorname{Re} = \frac{\rho v D_h}{u} \tag{1}$$

Table 1. Summary of Reynold Numbers

Velocity (m/s)	Reynolds Number	Classification	
0.5	13580.95	Turbulent	
2.5	67904.75	Turbulent	
5	135809.5	Turbulent	

The flow within the dryer was found to be turbulent for all three velocity cases. There are three different computational approaches that can be taken to model turbulent flow, these are direct numerical simulation (DNS), large eddy Simulation (LES) or Reynold Average Navier-stokes Simulation (RANS) [18]. Each approach has their own advantages and disadvantages for a particular application.

A RANS SST k- ω turbulence model was selected to be used as the model choice for the dryer. This model is considered to be a combination of a k $-\varepsilon$ in the free stream and k- ω near the walls. The turbulence model does not rely on wall functions and can be more accurate when solving complex flow near the walls of a geometry [19, 20]. A tray style batch dryer can have quite complex flow pattern around the trays and tray holders commonly creating dead spots and flow separations throughout the dryer. A SST k- ω turbulence model provides superior performance for wall bounded boundary layers and is considered to be more accurate in predicting flow separation over other RANS model [17]. The simulation will be steady state and solve the SST model as stationary. The Menter's SST turbulence model equation are formulated in terms of two variables, k and ω which can be written as,

$$\rho u \cdot \nabla k = P - \rho \beta_0^* k \omega + \nabla \cdot \left[\left(\mu + \sigma_k \mu_T \right) (\nabla k) \right]$$
⁽²⁾

And
$$\rho u \cdot \nabla \omega = \frac{\rho \gamma}{\mu_T} P - \rho \beta \omega^2 + \nabla \cdot \left[\left(\mu + \sigma_\omega \mu_T \right) (\nabla \omega) \right] + 2 \left(1 - f_{\nu 1} \right) \frac{\rho \sigma_{\omega 2}}{\omega} \nabla \omega \cdot \nabla k$$
 (3)

The definitions and expressions of these variables can be found in Menter's model [17].

4. Mesh

The meshing process for the geometry is key in any CFD analysis. As SST $k - \omega$ turbulence model doesn't rely on the wall functions, the mesh must be designed appropriately. A single mesh for each inlet/outlet case was used. This

resulted in two separate meshes being utilised, one for the configuration 1 and another for configuration 2. The final meshes used can be seen in Figure 2.



Fig 2. Final Mesh - Top View (a) Configuration 1 (b) Configuration 2

Mesh 1 is a patch conforming structured mesh made up of only hexahedron cells. The cells are aligned with the predominate direction of the flow to reduce numerical error [21]. The distribution of the cells favours the walls as required due to the SST k- ω turbulence model choice.

Due to the outlet location for configuration 2, the same mesh could not be utilised. This resulted in a slightly different mesh, where the bend portion of the domain was made entirely tetrahedrons. When hexahedron cells are not aligned with the predominate direction of the flow they lose their advantage over tetrahedron cells. From how the outlet is positioned within the geometry, it is assumed the air will be drawn up into the outlet causing the predominate direction of the flow cells being utilised within the geometry.

5. Boundary Conditions

In the simulations, three key boundary conditions namely, an inlet, outlet and wall treatment were used. The boundary conditions were kept constant for all the simulations, with only the positions and velocities changing for the appropriate simulation. A velocity inlet, pressure outlet with the wall treatment of no slip was used to aid in the convergence of the solution. These boundary conditions expressed as equations can be seen through equations (4) to (6).

$$V_{in} = v m/s \tag{4}$$

$$P_{out} = 0Pa \tag{5}$$

$$\mu = 0, \ k = 0, \ \omega = \frac{\lim_{l_w} 6\mu}{l_w \to 0} \frac{6\mu}{\rho\beta_1 l_w^2}$$
(6)

6. Results and Discussion

The results for the six simulation runs have been presented in Figures 3 to 6. Both configurations resulted in vastly different flow distributions. The results shown in Figures 3 to 5 are of a single slice through the centre of the geometry. Figure 6 shows five slices spread evenly throughout the geometry to show the full distribution.





Fig 3. Velocity Magnitude slice through the centre of the geometry (a) Configuration 1 - 0.5 m/s (b) Configuration 2 - 0.5 m/s



Fig 4. Velocity Magnitude slice through the centre of the geometry (a) Configuration 1 - 2.5 m/s (b) Configuration 2 - 2.5 m/s



Fig 5. Velocity Magnitude slice through the centre of the geometry (a) Configuration 1-5 m/s (b) Configuration 2-5 m/s



Fig 6. 5 Velocity Magnitude slices spread evenly throughout the geometry (a) Configuration 1 - 0.5 m/s (b) Configuration 2 - 0.5 m/s

Configuration 1 has a single inlet and outlet at each end of the U. Within its flow distribution the flow hits the inside wall, accelerating through the bend creating two dead zones within the geometry. One dead zone is located on the outside edge of the bend and the other is located along the inside edge of the wall next to the outlet. Figure 6 (a) shows the full flow distribution of configuration 1 at 0.5 m/s. In this figure the full extent of the dead zones can be seen. The dead zone on the inside edge next to the outlet is quite large with flow separation/recirculating regions occurring. Results show that the inlet velocity affects these dead zones; the faster the inlet velocity the greater/larger the dead spot and/or the greater the effect the bend has upon the flow distribution.

Configuration 2 has two inlets and a single outlet at the top centre of the bend within the U. This configuration has very different flow distribution than configuration 1. The position of the outlet minimises the effect the bend has upon the flow distribution however configuration 2 still has a greatly uneven distribution, as it can be seen in Figure 3 to 5 (b). This configuration has one dead spot below the centre of the outlet. The location of the outlet also created the air in the region to be accelerated towards the outlet, which was expected. These two issues occur where no trays/tray

holders are to be positioned within the geometry, which would allow these issues to have a reduced affect upon the product. Figure 6 (b) shows the full distribution on configuration 2 at 0.5m/s. This figure shows quite an even distribution however, it should be noted the scale within the figure is quite different with a maximum velocity from this simulation above 1.8 m/s. From figure 6 (b) it appears that configuration 2 would benefit from further simulation with an expanded domain to evaluate the full effect of the outlet position upon the airflow distribution.

7. Conclusion

Intermittent Microwave Convection Drying (IMCD) is an innovative concept that has the potential to reduce drying time and improve both energy efficiency and product quality. Two variables can greatly affect the airflow distribution within a dryer, its geometry and the arrangement of its inlets/outlets.

Both configurations simulated have their own positives and negatives, however both have relatively uneven distributions. Configuration 2 has one dead zone, which occurs where no tray/tray holders are located within the bend, with the flow distribution being only marginally affected by the bend in the geometry. Configuration 1 shows the second dead zone caused by the bend within the geometry, and is strongly affected by the inlet velocity of the air, which changed greatly depending upon the vertical position along that wall. This study shows how dead zones can be minimised by changing the location of inlets and outlets.

References

- [1] Australian Bureau of Statistics. Agricultural production; 2012, Accessed on 13 August 2016, Retrieved from http://www.abs.gov.au/ausstats/abs@.nsf/Lookup/by%20Subject/1301.0~2012~Main%20Features~Agricultural%20production~260.
- [2] Revised Queensland Science and Research Priorities. Office of the Queensland Chief Scientist; 2015, Retrieved from http://www.chiefscientist.qld.gov.au/images/documents/chiefscientist/pubs/reports-other/qld-science-n-research-priorities-2015-2016.pdf
- [3] Kumar C, Joardder MUH, Farrell TW, Millar GJ, Karim MA. Mathematical Model for Intermittent Microwave Convective (IMCD) Drying of Food Materials. Drying Technology 2015; 34(8): 962-973.
- [4] Kumar C, Joardder MUH, Farrell TW, Karim MA. Multiphase porous media model for intermittent microwave convective drying (IMCD) of food. International Journal of Thermal Science 2016; 104: 304-314.
- [5] Kumar C, Joardder MUH, Karim MA. Intermittent drying of food products: A critical review. Journal of Food Engineering 2014; 121: 48-57
- [6] Joardder MU, Brown RJ, Kumar C, Karim M. Effect of cell wall properties on porosity and shrinkage of dried apple. International Journal of Food Properties. 2015; 18(10):2327-2337.
- [7] Kumar C, Joardder MUH, Karim A, Millar GJ, Amin Z. Temperature redistribution modelling during intermittent microwave convective heating. Procedia Engineering. 2014; 90:544-549.
- [8] Joardder MUH, Karim A, Kumar C. Effect of temperature distribution on predicting quality of microwave dehydrated food. Journal of Mechanical Engineering and Sciences. 2013; 5:562-568.
- [9] Norton T, Sun D. Computational fluid dynamics (CFD) an effective and efficient design and analysis tool for the food industry: A review. Trends in Food Science & Technology 2006; 17: 600-620.
- [10] Misha, S, Mat S, Ruslan M, Sopian K, Salleh E. The CFD Simulation of Tray Dryer Design for Kenaf Core Drying. Applied Mechanics and Materials 2013; 393: 717-722.
- [11] Murathathunyaluk S, Srisakwattana N, Saksawad T. Development of Rotating Tray Dryer and Study of the Hot Air Flow Pattern with Computational Fluid Dynamics. Chemical Engineering Transactions 2015; 43:1-6.
- [12] Ghiaus A, Gavriliuc R. Simulation of Air Flow and Dehydration Process in Tray Drying Systems. U.P.B Science Bulletin 2007; 69(D):41-50
- [13] Amanlou Y, Zomorodian A. Applying CFD for designing a new fruit cabinet dryer. Journal of Food Engineering 2010; 101: 8-15.
- [14] Yongson O, Badruddin I, Zainal Z, Aswatha Narayana P. Airflow analysis in an air conditioning room. Building and Environment 2007; 42: 1531-1537.
- [15] Sufán F. Experimental And CFD Study On Low Temperature Drying Of Loose And Compressed Bulks. Dissertation 2014.
- [16] Amjad W, Munir A, Esper A, Hensel O. Spatial homogeneity of drying in a batch type food dryer with diagonal air flow design. Journal of Food Engineering 2015; 144:148-155.
- [17] Bergman T, Incropera F. Fundamentals of heat and mass transfer. 7th edition. Hoboken NJ: John Wiley and Sons; 2011.
- [18] ANSYS. Modelling Turbulent Flows Introductory Fluent Training; 2006, Retrieved from http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf
- [19] Walter Frei. Which Turbulence Model Should I Choose for my CFD Application?. COMSOL 2013, Accessed on 13 August 2016, Retrieved from https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/
- [20] Menter, F. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 1994; 32:1598-1605.
- [21] CAE ASSOCIATES. CFD Meshing with ANSYS Workbench; 2013, Retrieved from https://caeai.com/sites/default/files/CFD_Meshing_CAEA.pdf