Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 110 (2017) 465 – 470

Energy Procedia

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.

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

Peer-review under responsibility of the organizing committee of the 1st International Conference on Energy and Power.
doi:10.1016/j.egypro.2017.03.170

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
P	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.