IOPscience iopscience.iop.org

Home Search Collections Journals About Contact us My IOPscience

Simulation of Hybrid Solar Dryer

This article has been downloaded from IOPscience. Please scroll down to see the full text article.

2013 IOP Conf. Ser.: Earth Environ. Sci. 16 012143

(http://iopscience.iop.org/1755-1315/16/1/012143)

View the table of contents for this issue, or go to the journal homepage for more

Download details:
IP Address: 175.140.118.147
The article was downloaded on 15/07/2013 at 01:46

Please note that terms and conditions apply.

4th International Conference on Energy and Environment 2013 (ICEE 2013) IOP Publishing
IOP Conf. Series: Earth and Environmental Science **16** (2013) 012143 doi:10.1088/1755-1315/16/1/012143

Simulation of Hybrid Solar Dryer

Y M Yunus, H H Al-Kayiem[1]
Mechanical Engineering Department, Universiti Teknologi PETRONAS
[1]E-mail: hussain_kayiem@petronas.com.my

Abstract. The efficient performance of a solar dryer is mainly depending on the good distribution of the thermal and flow field inside the dryer body. This paper presents simulation results of a solar dryer with a biomass burner as backup heater. The flow and thermal fields were simulated by CFD tools under different operational modes. GAMBIT software was used for the model and grid generation while FLUENT software was used to simulate the velocity and temperature distribution inside the dryer body. The CFD simulation procedure was validated by comparing the simulation results with experimental measurement. The simulation results show acceptable agreement with the experimental measurements. The simulations have shown high temperature spot with very low velocity underneath the solar absorber and this is an indication for the poor design. Many other observations have been visualized from the temperature and flow distribution which cannot be captured by experimental measurements.

1. Introduction

Computational fluid dynamics (CFD) software could be used in solar drying field to predict air velocities and temperature distribution which is in manually will require a lot of sensors. Besides, the CFD may be used as a drying optimization tool to improve the design and to predict the drying time if connected to the thin-film equation [1]. [1] simulated a dryer which was designed to dry fruits and vegetables, consist of heat exchanger to enhance the heated air supplied from LPG gas burner. Centrifugal fan was located inside the dryer chamber to force the air movement. They found two areas of low air velocity under different trays location.

[2] have used CFD method to analyze the airflow and heat transfer by varying positions of a fan in a solar dryer. They had identified that the second position produced more uniform temperature and air flow. The maximum airflow was observed at the inlet and reduced as it approaches outlet. The simulation model described the real condition up to 89%.

[3] studied the heat and mass transfer in a solar dryer with biomass backup burner to obtain the optimum operating temperature using CFD software, STAR-CD. The modeling was performed on an empty chamber without the pepper berries. The simulation had been conducted under natural and forced convection. They had identified that heat and mass transfer by natural convection is more suitable for drying pepper berries with solar radiation. The simulation results indicate that the thermal heat distribution in the drying chamber was uniformly distributed.

There are no precise references to the literature concerning numerical simulation of air flows and heat transfer in the interior of dryers. However, there were shortages in the literature for the attempts of dryer simulation. The studies that had been conducted by previous researchers have shown that the CFD tools are very flexible to generate result close to the real processes.

The main objective of the present work is to investigate the drying performance of hybrid solar dryer using thermal backup by CFD simulation. The results are to be presented in form of velocity and temperature contours to visualize the flow and thermal fields' distributions.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd