Simulation of Hybrid Solar Dryer

Y M Yunus, H H Al-Kayiem
Mechanical Engineering Department, Universiti Teknologi PETRONAS

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.
2. Computational implementations

A 2-D model of solar dryer geometry was created using GAMBIT version 2.2.30. The inlet source for solar and biomass modes were different according to the type of the operational mode. The solar drying mode was design with air inlet located near the absorber plate at the side wall. In order to maximize the heat received from the back up burner and to minimize the hot air escape to the outside, the air inlet was closed in biomass mode. Label of the model is shown in figure 1.

![Figure 1. Labeled GAMBIT dryer model.](image)

2.1 Computational Grid

The domain was discretized into a finite set of control cells. It was meshed with tri mesh elements and the face meshing type was pave, meaning an unstructured grid. The bottom part of the model was meshed with smaller interval size of 1.5 in order to get more accurate results surrounding the solar collector and biomass burner inlet. The middle and upper parts of the dryer were meshed with larger interval size of 2 and 3 to reduce the simulation time.

2.2 Governing Equations of Thermo Fluid Process

The governing equations for CFD are based on conservation of mass, momentum, and energy. FLUENT uses a finite volume method (FVM) to solve the governing equations. The FVM involves discretization and integration of the governing equation over the control volume. Several assumptions have been made for the solar dryer model. The design model is in two dimensional 2-D, the flow is in x and y-direction, steady, and compressible flow. Since the working fluid is gas, the body forces are very small and could be neglected.

2.3 Simulation procedure

**Radiation Model:** P-1 radiation model had been used. The radiation flux, \( q_r \), is given by (1).

\[
q_r = \frac{1}{3(\alpha + \sigma_t) - C\sigma_t} \nabla G
\]

(1)

where, \( \alpha \) is the absorption coefficient, \( \sigma_t \) is the scattering coefficient, \( G \) is incident radiation and \( C \) is the linear-anisotropic phase function coefficient.

**Turbulence Modeling:** In this study, a turbulence model of two equations, standard \( k-\varepsilon \) model [7] had been used for the case of biomass burner mode. The turbulence model for mode that involves radiation was modeled using Renormalization Group (RNG). It is based on the two equation models of the Reynolds-Averaged Navier Stokes (RANS) equations. This RNG \( k-\varepsilon \) model is able to produce more accurate results than the standard \( k-\varepsilon \) model since it has an additional term in its \( \varepsilon \) equation.
Boundary Conditions: The boundary conditions of the model were divided into two categories, which were solar mode and biomass mode. The boundary conditions of both modes were consisted of six zones. In solar mode only, the burner was disconnected and left as the air inlet. The thermal condition is depending on the type of model. The thermal conditions with and without P1 model were set as mixed and convection respectively. Further details are available in [8].

3. Results and discussion
3.1 Numerical Solution
Within this CFD simulation, the equations for: Flow, Turbulence, Energy and Radiation were solved. The discretization and solution schemes used for these equations are: Semi Empirical Pressure (SIMPLE), Standard and First Order Upwind scheme for Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate and Energy. Convergence was monitored by checking the information from computed residuals of all equations. Scaled residuals were used. Computations were continued until the values of residuals were progressively reducing by typically five or six orders of magnitude, although in a few cases this could not be achieved. The calculations were considered convergent if the scaled residual for the continuity equation, the energy equation, P-1 radiation was less than $10^{-3}$, $10^{-8}$ and $10^{-6}$ respectively.

3.2 Analysis of Results
The results were focusing on the airflow and temperature distribution inside the dryer. It was presented according to the mode of drying from the unloading experimental results. The validation of the result was done by simulating the initial operating conditions of experimental measurement. The percentage of error was calculated by finding the difference between the experimental and simulated result.

The velocity vectors of air flow inside the solar dryer under solar mode are shown in figure 2. Ambient air flows into the dryer through the side opening holes at the base of the solar dryer and the front holes near the collector. The maximum velocity that can be achieved is 0.26 m/s and it is found to be concentrated at small path in the middle of collector. The air flows to the drying chamber and when it reaches small path of first tray, some of the air flow hits the bottom face of the trays causes a backflow to occur. This is similar to the results reported by [1]. The velocity increases as it flows through the alternate holes gap of the trays across the drying chamber and finally out at chimney. From simulation results, mean velocity of air escaping from chimney is shown to be 0.17 m/s. The value is approximately same as the velocity obtained through experiments which was 0.19 m/s. Observation of the velocity distribution indicated that the flow velocity in the solar dryer is not uniform. The flow velocity is high mainly at the gaps of the trays.

The temperature distribution inside the solar dryer is shown in figure 3. It could be seen that the highest temperature is concentrated surrounding the solar absorber plate which is around 330K to 341K. The heat was transferred from bottom to the top part of the dryer by natural convection. The first and second tray recorded the highest temperature around 318K since it is exposed to the high temperature air flow. However, as the location of the tray become farther, the temperature decreases due to the heat losses to the surroundings and to heat up the lower trays. The lowest temperature was spotted below the absorber plate and it represents the ambient air temperature which flow from the side walls of dryer. The simulation results indicated that the heat convection and heat conduction received from the absorber plate and walls respectively maintains the drying chamber at high temperature.
The comparison between the simulation and experimental results is shown in Table 4. The results show that temperature near the absorber plate is in good agreement since the difference is around 11%. The measured values of the temperatures are higher than the simulated value due to the presence of solar radiation. The highest difference is at tray 5 since it is located at the top of drying chamber; hence it has the largest area exposed to the sun. In addition, the setting of P1 radiation heat transfer at the wall of dryer in is simulation is inadequate as compared to real condition.

| T\text{near collector}(°C) | Tray 1 (°C) | Tray 3 (°C) | Tray 5 (°C) |
|---------------------------|-------------|-------------|-------------|
| Experimental              | 66.5        | 59.8        | 53.8        | 65.8        |
| Simulation                | 59.0        | 45.1        | 43.4        | 45.0        |
| Difference (%)            | 11.3        | 24.6        | 19.3        | 31.6        |

Conclusions

The simulations gave a reasonable result to real conditions. The simulation results were validated by comparing with the results obtained from experiment measurements. The results indicate that the calculated percentage of errors in thermal backup and hybrid modes were in good agreement. Hybrid mode is the most appropriate mode for drying application since it provides the highest temperature and falls within the required drying temperature range. The low temperature spot was identified and will be assisted in the improvement of dryer design. The simulation results provide practical information by identifying the weakness of the system and consequently assist in design improvement.

[1] Mathioulakis E, Karathanos V T, and Belessiotis V G 1998 J. of Food Eng 36(2) 183.
[2] Dyah W, Nelwan L O, Kamaruddin A, and Indra S A 2003 IPB (Bogor Agricultural University) 17(1).
[3] Rigit A R and Low P T 2010 WASET 115-118.
[4] Bird R, Stewart W, and Lightfoot E Transport Phenomena New York: John Wiley & Sons Inc.
[5] Incropera F P, Dewitt D P, Bergman T L, and Lavine A S 2007 5\textsuperscript{Ed}. John Wiley & Sons.
[6] Vest C M 1979 Holographic interferometry New York: John William & Sons.
[7] Launder B and Spalding D, 1974 Computer Methods in App. Mech. and Eng 3 269-289
[8] Md Yunus Y 2011 MSc thesis Universiti Teknologi PETRONAS.
http://utpedia.utp.edu.my/3060/