Thermal Analysis on Heat Pump Chili Drying Chamber with variations in the direction of air flow using Computational Fluid Dynamics

A H Siburian\textsuperscript{1,2}, P G Silalahi\textsuperscript{1,2} N D M Romauli\textsuperscript{3} and H Ambarita\textsuperscript{1,2*}

\textsuperscript{1}Mechanical Engineering Department, Universitas Sumatera Utara, Medan 20155, Indonesia
\textsuperscript{2}Sustainable Energy and Research Centre, Universitas Sumatera Utara, Medan 20155, Indonesia
\textsuperscript{3}Food Science and Technology, Faculty of Agriculture, Universitas Sumatera Utara, Medan 20155, Indonesia

*himsar@usu.ac.id

Abstract. Drying chilies using a heat pump includes drying at low temperatures so that the temperature distribution and flow direction in the drying chamber need to be considered. The distribution of temperature and flow direction in the heat pump drying room with a drying room size of 1 m x 0.8 m x 0.75 m to dry 1 kg of chilies built in the Sustainable Energy and Research Center (SERC) laboratory is not good enough to make the drying time longer. The procedure has been established with the aim of evaluating the speed distribution and temperature of the drying air in the drying chamber to determine the need for redesign. Computational Fluid Dynamics (CFD) is used to analyze flow patterns that occur in the drying chamber where in this study two types of flow directions that enter the heat pump chili drying chamber are compared. CFD has replaced the classical numerical analysis method of drying processes based on experimental models. Analytical methods and experimental methods are limited in describing the distribution of temperature and air flow in the drying chamber. In this case, the finite element method is used in the form of CFD analysis where the geometry of the drying chamber is divided into several analyzed elements. After doing the CFD simulation, it is obtained that the heat pump chili drying chamber model with the flow direction from below the temperature increases faster than the heat pump chili drying chamber model with the flow direction from the side. The simulation results show that the drying room with the direction of hot air flow from below, the temperature of the dried chilies increases faster and the temperature distribution is better than the drying room with the flow direction from the side. In both models of the drying chamber, the temperature value is obtained in the timestep 600, namely in the drying chamber model with hot air flow from the side is 305,569 K and in the drying chamber model with hot air flow from below is 313,697 K. For the drying model in this heat pump chili drying room it is advisable to use the flow direction from below because it is better. This solution can also be offered to get the right drying chamber model.
1. Introduction

Heat pump drying is a drying device that utilizes an evaporator and condenser to condition the air used to dry the dried material. Air has low humidity as it passes through the evaporator and has an increased temperature as it passes through the condenser. Among the equipment available in the drying industry, heat pump dryers are the most widely used, which have relatively high heat recovery potential and energy efficiency [1]. The drying techniques were classified as cheap drying techniques such as sun drying and oven drying and also expensive drying techniques such as vacuum drying and freeze-drying is depend on oven market price [2]. Drying is not only applied to the food sector, but also extends to the biochemical industry such as pharmaceuticals and the agricultural sector [3]. Foodstuffs produced from agricultural products contain water content which can affect the physical condition of foodstuffs. However, the characteristic and color of variety are not changed when through drying process such as seaweed cultivars is one of variety which found in fishing communities in Indonesia [2]. Besides of seaweed cultivars, drying process is important to implement in agricultural commodity such as Chili (Capsicum L annuum L) that is widely cultivated in Indonesia. In recent years, much effort has been made to design and develop many chili dryers. These dryers can be classified in natural and forced convection [4]. Temperature and air velocity are two factors that must be considered as they are difficult to measure due to the large number of sensors that must be placed in the room. The results of the drying process depend on the location of the material in the dryer, because the drying rate depends on the air flow entering the drying chamber [6]. Powerful computational tools such as Computational Fluid Dynamics (CFD) have now replaced the classical method of numerical analysis of drying processes based on experimental models [7].

The purpose of this study is to obtain a good drying chamber model using the three-dimensional Computational Fluid Dynamics (CFD) model of the heat pump drying chamber built. Analysis of 2 drying chamber models on the behavior of air flow in the geometry of the drying chamber and its effect on temperature distribution during the convective drying process.
2. Method

2.1. Research Methods

![Research Flow Diagram](image)

**Figure 1** Research Flow Diagram

Figure 1 illustrates the experiment flow. First, create the geometry of the object being analyzed using ANSYS Design Modeler. Second, create a geometric mesh and enter the boundary conditions of the simulation. The input data is air temperature and air velocity entering the drying chamber. After that,
start the simulation with the required time limit until the iteration calculation is complete. And then graph, analyze and discuss.

2.2 CFD Governing Equation Model

There are several important PDE (Partial Differential Equations) that must be solved to determine the transfer of heat, mass and momentum during the drying process. Newtonian fluid flow modeling uses the Navier-Stokes equation. The drying process involves heat transfer and fluid properties that are temperature dependent, so the energy equation is usually combined with the Navier-Stokes equation. Mathematically, CFD replaces the partial differential equations of continuity, momentum and energy with linear algebraic equations. In this case, it is analyzed under 3-dimensional conditions. CFD is an approach from a problem whose origin is a continuum (having an infinite number of cells) to a discrete model (finite number of cells). The laws of physics that explain fluid flow and temperature distribution are the law of conservation of mass, law of conservation of momentum, law of conservation of energy.

2.1.1 Law of Conservation of Mass
The rate of mass increase in the fluid element = the net rate of mass flow into the finite element with a mathematical form [8]

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\] (1)

Where is \( \rho \) the density (kg/m\(^3\)), \( t \) is time (s), x,y,z is Cartesian coordinates (m), u, v, w is the component of speed (m/s).

2.1.2 Momentum Equations
The momentum equation is a Navier-Stokes equation in the form corresponding to the finite volume method [8]:

Momentum x direction :

\[
\rho \frac{Du}{Dt} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx}
\] (2)

Momentum y direction :

\[
\rho \frac{Dv}{Dt} = \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My}
\] (3)

Momentum z direction :

\[
\rho \frac{ Dw}{Dt} = \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial (-p + \tau_{zz})}{\partial z} + S_{Mz}
\] (4)

2.1.3 Energy Equations

\[
\rho \frac{DE}{Dt} = -\text{div}(pu) + \left[ \frac{\partial (u \rho u)}{\partial x} + \frac{\partial (u \rho v)}{\partial y} + \frac{\partial (u \rho w)}{\partial z} \right] + \text{div}(k \text{grad}T) + S_{E}
\] (5)

2.1.4 Reynold’s Number Equation

\[
\text{Re} = \frac{\rho v D_h}{\mu}
\] (6)
2.1.5 Internal Turbulent Inner Flow equation

\[ \text{Nu}_D = 0.023 \text{Re}^{0.8}_D \text{Pr}^{0.4} \]  \hspace{1cm} (7)

2.1.6 Convection heat transfer formula

\[ Q_{\text{air}} = hA(T_1 - T_2) \]  \hspace{1cm} (8)

2.3 Drying Chamber Modeling

The design of the heat pump chili dryer in the SERC laboratory is shown in Figure 2. The dynamics of computational fluids have developed rapidly in industrial applications and academic research [12]. CFD modeling can help reduce design modeling significantly thereby reducing modeling costs and processing time. The geometric design is based on the size data of the heat pump chili drying room, the dimensions are 1 m x 1 m x 0.75 m, the walls and roof are composed of 2 layers of aluminum with a thickness of 0.001 m, and limited by 0.02 m thick cork. The CFD simulation was developed in the ANSYS FLUENT 2020 R1 Academic Version software.

![Figure 2. Heat pump chili dryer design](image)

The geometry of the heat pump drying chamber was created using the ANSYS Design Modeler program. Geometry is made at a scale of 1: 1. The geometry created is a drying chamber, drying rack and chilies. In this simulation, 2 drying chamber models are varied as shown in Figures 3 (a) and (b).

![Figure 3. Drying chamber model (a) model 1 (b) model 2](image)
Creating a mesh or grid is an important consideration in achieving numerical solutions for the partial differential equations that govern this CFD case [4]. The mesh generation is carried out in stages, namely creating a mesh on the overall geometry as in Figure 4 then inflating the conditions of the shelf and chili boundaries as well as the drying chamber divider with a maximum layer is 3 and a Growth rate is 1.2.

![Figure 4. Mesh](image)

Table 1 and 2 show the skewness value and the drying chamber model parameters, while Table 3 and 4 contains the boundary conditions and material for the phenomena that occur. Inlet and outlet spatial discretization is modeled as velocity inlet and pressure outlet and discretization residue is modeled as wall condition. The selection of the physical model in this case is shown in table 1 and is modeled as a transient system with turbulent flow because it is through the calculation of the Reynold number of inner flow with a value is 25442.53. The required parameters specified in the model are the time step, iterations in per time step and the maximum physical time. The most important parameter is the time step, because if it is not calculated correctly, various problems can arise. One of these problems is convergence, which occurs when the time step is greater than the velocity.

| Table 1. Number of grids and skewness in the drying chamber model |
|---|---|---|---|
| No | Model | Number of Grids | Skewness |
| 1 | Drying Chamber | 179404 | 0.86 |

| Table 2. Model parameters |
|---|---|
| No | Parameter | Information |
| 1 | Energy | On |
| 2 | Viscous | K- e Realizable |

| Table 3. Drying Chamber boundary conditions |
|---|---|---|
| No | Parameter | Information |
| 1 | Velocity Inlet | 2.6 m/s |
| 2 | Initial Drying Chamber Temperature | 299.6 °K |
| 3 | Initial Chili temperature | 299 °K |
| 4 | Operating Pressure | 1 atm |
Inlet Air Temperature 327.7 °K

Table 4. Materials

| No | Material | Density (kg/m³) | Specific Heat (J/kg °K) | Thermal Conductivity (W/m K) |
|----|----------|----------------|------------------------|----------------------------|
| 1  | Aluminium | 2719           | 871                    | 202.4                      |
| 2  | Chili     | 500            | 4170                   | 0.49                       |
| 3  | Cork      | 120            | 1800                   | 0.039                      |
| 4  | Glass     | 2500           | 750                    | 1.4                        |
| 5  | Zinc      | 7140           | 389                    | 116                        |

3. Results and Discussions
The results of the speed profile can be seen in Figure 5. The results of the chili and drying chamber temperature profiles can be seen in Figures 6 and 7. The chili temperature in the drying chamber model 2 in Figure 6 (b) is higher in the same drying simulation time than the drying chamber model 1 in Figure 6 (a).

![Figure 5](image-url)  
(a) Vector of airflow velocity (a) model 1 (b) model 2
Figure 6. The temperature distribution in the drying chamber model (a) model 1 (b) model 2. The drying chamber model for model 1 in Figure 6 (a) has an air flow that flows from the side where the hot air flow is evenly distributed to the drying chamber but less well distributed on the rack and dried chilies. The drying chamber model 2 in Figure 6 (b) has an air flow that flows from below where the hot air flow is evenly distributed to the shelves and chilies.

Figure 7. Temperature contour of chilies (a) model 1 (b) model 2

From the simulation, it is found that model 2 of the drying chamber in Figure 7 (b) with the direction of hot air flow from below, the temperature rises faster and the temperature distribution in chilies is better than the direction of air flow from the side of model 2 in Figure 7 (a). In both models of the drying chamber, the temperature value is obtained in the timestep 600, namely in model 1 of 305,569 K and in model 2 of 313,697 K. Gómez et al [5] conducted a simulation study of temperature distribution. To understand the convective drying process, a CFD simulation is performed using a transient system model with K-e Turbulent and CFL criteria where there are two specific areas with low velocity that tend to cause recirculation. Simulations with CFD are demonstrated as reliable optimization tools to avoid expensive experiments to refine designs.
In the drying chamber section of model 1 in figure 8 (a) more heat is transferred to the drying wall than model 2 in figure 8 (b). In the drying chamber model 1 more heat is wasted into the drying walls than model 2. The temperature distribution and flow direction in model 1 and model 2 are in accordance with those expected where the temperature distribution and flow direction are acceptable. The drying chamber model 2 is better so it can be suggested as a drying room model for the heat pump chili drying chamber designed in the SERC laboratory.

4. Conclusions
Computational Fluid Dynamics simulations were carried out for 2 drying chamber models using a transient system model and turbulent flow to obtain a heat pump chili drying chamber model with good drying. In this CFD analysis using the finite volume method approach. The distribution of temperature and air flow in model 2, the airflow from below is better than in model 1, the airflow from the side. The temperature distribution in model 2 increases faster and evenly distributed than model 1. Simulations using CFD can be used as a means of obtaining an effective drying chamber model without expensive experiments to optimize existing designs or create new designs. The results of this study indicate that in the drying chamber model the air flow should be adjusted to flow from below so that drying is more effective and the temperature distribution is evenly distributed.

References
[1]. Tong Y, Kozai T, Nishioka N, and Ohyama K 2010 Greenhouse heating using heat pumps with a high coefficient of performance (COP) Biosyst. Eng. 106 4, pp. 405-411
[2]. Charles A L, Sridhar K and Alamsjah M A 2019 Effect of drying techniques on color and bioactive potential of two commercial edible Indonesian seaweed cultivars J. of Applied Phycology 32 1 pp 563-572
[3]. Mujumdar A S 2004 Research and development in drying: Recent trends and future prospects Dry. Technol. 22 1 pp. 1-26
[4]. Gupta P, Ahmed J, Shivhare U S and Raghavan S V 2002 Drying Characteristics of Red Chilli, Drying Technol. 20 10 pp. 1975-1987
[5]. Miguel Andrès M D, Carlos A Gomez, Ratkovich N and Gómez Daza J C 2018 Numerical Analysis of a Convective Drying Chamber from Drying Air Velocity and Temperature Perspective. Proc. of the 3rd World Congress on Momentum, Heat and Mass Transfer (MHMT’18) 134
[6]. S VijayaVenkataRaman, S Iniyan, and Goic R 2012 A Review of Solar Drying Technologies, *Renewable and Sustainable Energy Reviews* **16** pp. 2652-2670

[7]. Ramachandran R P, Akbarzadeh M, Paliwal J, and Cenkowski S 2017 Computational Fluid Dynamics in Drying Process Modelling-a Technical Review *Food and Bioprocess Technology* **11**

[8]. Versteeg H K and Malalasekera W 2007 *An Introduction to Computational Fluid Dynamics. The finite volume method* Second Edition. Pearson Education Limited. England.

[9]. Massoud, M 2005 Engineering Thermofluids: Thermodynamics, Fluid Mechanics, and Heat Transfer (Springer. Germany)

[10]. ANSYS, Inc. ANSYS Workbench User’s Guide 2020 R1. Southpointe 2600 ANSYS Drive Canonsburg PA 15317; USA, 2020.

[11]. Ambarita H. 2017. Perpindahan Panas dan Massa. Penyelesaian Analitik dan Numerik. Intelegensia Media. Malang, Indonesia

[12]. Tu J, Yeoh H Y, and Liu, C 2018 *Computational Fluid Dynamics. A practical Approach* Third Edition. (Elsevier. The Boulevard, Langford Lane, Kidlington, Oxford)