CFD Analysis of Temperature Drying Chamber at Rotary Dryer With Combined Energy

A Rindang*, S Panggabean1, F Wulandari1

1Department of Agricultural Engineering, Universitas Sumatera Utara, Prof. A. Sofyan No. 3 Kampus USU, Medan, Indonesia Republic-20155

*adian.rindang@usu.ac.id

Abstract. A heat and mass transfer model has been developed using Computation Fluid Dynamics (CFD) to see the pattern of hot air flow, in the drying chamber in the rotary dryer. During the experiment parameters used as input data on CFD are air flow velocity and temperature drying in experimental measurements in the range of 1 hour, 2 hours and 3 hours. CFD analysis results show the uneven distribution of heat in the drying chamber in a variation of 1 hour and 2 hours. The CFD contour indicates that heat only spreads to the top and center of the drying chamber. The average temperature of the top of the drying chamber is 51.75°C, while at the bottom is 45.19°C. The simulation results on 3 hours of drying showed that the contour of hot air spread fairly evenly in the drying chamber with an average drying room temperature of 46.3°C. The air flow rate and air velocity in the drying chamber are quite constant with an average of 3.3 m/s.

1. Introduction
One of the conditions for achieving good drying is the uniformity of the final water content of agricultural products. The direction of the hot fluid flow greatly affects the efficiency and uniformity of the final water content [1]. Uniformity is obtained if the heat distribution of the liquid is spread evenly and the drying air can be dispersed precisely in the dryer. The results in water content in the grain will yawn and carried away by flow air exit [2]. To get high efficiency in drying, things to note are air temperature, fluid velocity, and fluid distribution in the drying chamber the pattern of air flow in the drying chamber can be known by using a Computational Fluid Dynamics (CFD) simulation [3]. This simulation makes it possible to provide detailed information about heat distribution and mass transfer currents and makes it possible to evaluate geometric changes in the dryer [4]. There are many studies based on CFD simulations for various agricultural products, such as using CFD for industrial scale rack type dryers, the results obtained are the level of fruit dryness depending on the position in the dryer [5]. When determining the pressure profile and air velocity with CFD, it shows that the main causes of various levels of drying and humidity are the lack of spatial homogeneity of the air density in the dryer. Also studied the field of the speed of modern sausage dryers to provide information about air circulation in the drying chamber [3]. The main objective of this research is to look at the pattern of heat flow distribution and the amount of temperature scattered in the drying chamber.

2. Materials and Method
2.1 Experimental Design
This research was carried out in the Laboratory of Biosystem Engineering, Faculty of Agriculture and Laboratory of Mechanical Engineering, Faculty of Engineering, Universitas Sumatera Utara, Medan, Indonesia. A rotary dryer, heater, hand anemometer, DS18B20 temperature sensor, DHT22 RH sensor, data logger, thermometer, scales, and oven are used in this research. The main research object used is 35 kg of unhulled rice, ciherang variety drying over 3 hours. The 5.6 kg of rice
husk used as biomass fuel to generated heat also with the heat elements. The rotary dryer prototype as shown in Figure 1.

**Figure 1** The rotary dryer prototype

Information:
1. Blower: 3 inches; 220 Volt; 2 A; 3000/3600 rpm.
2. Drying chamber: a cylindrical shape with stainless steel material.
3. Furnace and heat exchanger chamber: in the form of a beam with a size of 43 x 43 x 30 cm, made of an iron plate with a thickness of 4 mm.

The drying chamber uses heating directly and indirectly. Direct heating uses heating elements placed on the drying chamber wall totaling 2 units, while indirect heating uses heat generated from the burning of biomass fuel husk furnace. The heat generated from the burning of rice husks is captured by a series of galvanized pipes that function as heat exchangers (HE), then the outside air is drawn into the rotary dryer using a centrifugal fan which is then heated in HE, then forced into the drying chamber. To avoid heat loss, the drying chamber is covered with glass wool. The velocity of air flow is measured by a hand anemometer. The position of the sensors are explained in Table 1.

| Sensor Code | Amount | Location                          |
|-------------|--------|-----------------------------------|
| 01          | 1      | The channel of wet unhulled rice feed |
| 02          | 1      | Channel of hot air                 |
| 03          | 1      | The channel of dried unhulled rice  |
| 04          | 1      | Channel of moist air               |
| 05-08       | 4      | Around the base of the drying chamber |
| 09-12       | 4      | Around the center of the drying chamber |
| 13-16       | 4      | Around the top of the drying chamber |

2.2 CFD Simulation
The making of the simulation includes a series of stages that begin with the collection of technical data and ends with the presentation of the simulation results in the form of contour plots and the appearance of air flow in the rotary dryer [4].

The drying model is proposed to predict humidity and temperature profiles based on CFD simulation follows the stages: variable initialization (water content for grain, temperature and absolute humidity of the drying agent); adjust or update values variable in time, for the water content of the grain and humidity at the limit layer on the surface of the grain; the calculation of the term source of conservation mass and energy equation [6].

Technical data collection is carried out as the first step before simulating. Searching for a number of data from various sources is done as a database requirement in solid work [7]. The data sought include the parameter values of the dry air fluid, the drying material of the drying chamber. The parameter values needed are specific heat ratio (cp/cv), molecular weight (kg/mol), dynamic viscosity (Pas), heat type (J/kgK), thermal conductivity (W/mK), and density (kg/m³) [8].

This heating process is convection hence, the energy produced by the heating furnace can be calculated using equation 1:
\[ Q = h \ A \ (T_1 - T_2) \]  
\[ (1) \]

With:
- \( h \): convection heat transfer coefficient (W/m\(^2\).K).
- \( A \): area (m\(^2\)).
- \( T_1 \): temperature inside the cylinder (\(^\circ\)C).
- \( T_2 \): inlet temperature (\(^\circ\)C).

The calculation of the Reynolds value used as input fluid flow profile data is done using equation 2:
\[ Re = \frac{\rho \ V \ D}{\mu} \]  
\[ (2) \]

With:
- \( V \): Fluid speed (m/s).
- \( \rho \): Mass of fluid type (kg/m\(^3\)).
- \( D \): Pipe diameter (m).
- \( \mu \): Fluid dynamic viscosity (Ns/m\(^2\)).

CFD is software that uses numerical analysis with volume control as an element of integration of equations, which consists of mass equilibrium, momentum, and energy equations [5].

The law of mass balance:
\[ \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) + \frac{\partial}{\partial z} (\rho w) = 0 \]  
\[ (3) \]

3-dimensional momentum equation:

Momentum direction x:
\[ \rho \left( \frac{\partial u}{\partial x} + \nu \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \]  
\[ (4) \]

Momentum direction y:
\[ \rho \left( \frac{\partial v}{\partial x} + \nu \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \]  
\[ (5) \]

Momentum direction z:
\[ \rho \left( \frac{\partial w}{\partial x} + \nu \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \]  
\[ (6) \]

The simulation of the grain drying process uses CFD simulation with a model for that hot and mass transfer product, which is implemented into Ansys-Fluent software. Model this can also be used for simulations from the drying process to other fruits, it is enough to change the thermal and physical properties product and initial contour conditions [9].

After the simulation is complete, the results are analyzed so that the temperature distribution information that occurs during the drying process can be known. The resulting airflow data is assumed to be steady. Validation the simulation model is done by comparing the measurement data directly and the data of CFD simulation prediction. It is characterized by the coefficient of determination (R\(^2\)) if a value close to 1 then the CFD simulation is approaching the actual results [10].

3. Results and Discussion

An important factor that must be considered in the drying system of agricultural products to maintain quality and quality, one of which is temperature, this is because the temperature affects the process of product respiration. Drying temperature greatly determines the rate of evaporation of water during the drying process [8]. If the temperature difference between the heating medium and the material to be drained is large, the speed of moving heat into the material will also increase. The temperature distribution that occurs during the drying time is shown in Table 2. The average inlet temperature is 62.88\(^\circ\)C, while the average temperature of the drying cylinder wall is 44.55\(^\circ\)C.

Figure 2. shows the results of CFD simulation in 1 hour, 2 hours and 3 hours of drying time respectively. The results of the three contours, shown that there is an increase in temperature in the drying chamber. However, the distribution of hot air spreads unevenly. Figure 2a is a simulation result for 1-hour drying process, indicating that the warmest temperature is found at the base of the drying chamber cylinder, this is because the area is right next to the hot air inlet. In 2 hours of the drying process, the distribution of hot air covers the base and center of the cylinder (Figure 2b). Then, after 3 hours, the hot air has spread evenly throughout the drying cylinder chamber as shown in Figure 2b. The average temperature at 1 hour, 2 hours and 3 hours is 48.49 \(^\circ\)C, 49.27\(^\circ\)C, 50.44\(^\circ\)C respectively. In the
other words, the temperature in the drying chamber has a linear correlation to the simulation time, it will increase due to several factors such as the temperature of the air entering through ventilation, the friction of grain products with the drying wall, respiration which still takes place from grain products.

**Graph 1** Temperature distribution during experimental measurement

![Graph 1](image)

**Figure 2** Temperature contour at 1 hour (a), temperature contour at 2 hours (b) and temperature contour at 3 hours (c)

Figure 3 is the CFD simulation of air flow velocity in the cylinder drying chamber that passes the outlet. From simulation in 1 hour, 2 hours and 3 hours the air flow velocity are 3.3 m/s, 3.2 m/s and 3.3 m respectively. Figure 3 is also shown the air flowing in the drying system passes through the top and center of the chamber than move to the bottom of the drying chamber.
Validation of the CFD models is done by comparing the air temperature data from the experimental measurement with the prediction result from the models. Validation was carried out 3 times at different times, at 1 hour, 2 hours, and 3 hours. Validation results indicate that the value of the dryer temperature predicted by the model approaches the actual temperature value. This is indicated by the coefficient of determination ($R^2$) as shown in Figure 4.

![Diagram](image)

**Figure 4** Validation of CFD models

4. Conclusion

Hot air in the cylinder of the drying chamber can spread evenly for 3 hours of the drying process. CFD simulation results are quite accurate in predicting the temperature distribution that occurs during the drying process. There is a good correlation between the experiment and the simulation, while the coefficient of correlation $R^2$ is 0.958, 0.838, and 0.952 (Fig.4) for variations of time 1 hour, 2 hours, and 3 hours of drying respectively. New research strategies are needed in order to improve the accuracy of the heat and mass transfer model, position of HE, centrifugal fan and inlet need to be reviewed so that the optimal distribution of hot air in the drying chamber can be achieved, taking into account the specific internal structure of the grain and its implementation into the simulation. The experimental results were
in agreement with the results predicted by the CFD simulation of the drying process. The images produced by the CFD simulation for the moisture and temperature distribution in the grain cross-section may be used for uniformity degree estimation of drying.

5. Acknowledgments

The research was funded by the Research Central of University Sumatera Utara, Medan for Internal Grand Research (TALENTA 2018) Number: 42/UN5.2.3.1/PPM/KP-TALENTA USU/2018 at 16th March 2018.

References

[1] Nursyamsi 2012 Membentuk Karakter Peserta Didik Melalui Proses Pembelajaran oleh Guru Kelas di MI/SD J. Tarb. al-Awlad 4 1 389–397
[2] M. Z & S. D. I 2014 Characteristics of Grain Quality, Physical Quality, and Milling Quality Rice Galur Harapan Rice Paddy 3
[3] P. S. Mirade & J. D. Daudin 2000 A Numerical Study of the Airflow Patterns in a Sausage Dryer Dry. Technol. 18 1 81–97
[4] X. Z, Z. F, W. N & L. X CFD 2012 Modeling and Simulation of Superheated Steam Fluidized Bed Drying Process International Conference on Computer and Computing Technologies in Agriculture
[5] J. I. Perén, T. van Hooff, B. C. C. Leite & B. Blocken 2015 CFD Analysis of Cross-Ventilation of a Generic Isolated Building with Asymmetric Opening Positions: Impact of Roof Angle and Opening Location Build. Environ 85 4 263–276
[6] W. F, S. H, M. J, K. H & G. A 2011 Increase Of Homogenity And Energy Efficiency Of Mixed-Flow Grain Drying Internationale Konferenz Land. Technik AgEng 137–143
[7] J. Wu, H. Zhang & F. Li 2017 A Study on Drying Models and Internal Stresses of the Rice Kernel During Infrared Drying Dry. Technol. 35 6 680–688
[8] R. P. Singh & D. Heldman 2014 Introduction to food Engineering ed 5 (London: Elsevier)
[9] R. Ramponi & B. Blocken 2012 CFD Simulation of Cross-Ventilation for a Generic Isolated Building: Impact of Computational Parameters Build. Environ. 53 34–48
[10] D. Zare & M. Ranjbaran 2012 Simulation and Validation of Microwave-Assisted Fluidized Bed Drying of Soybeans Dry. Technol. 30 3 236–247