



Analysis of Inlet Temperature and Airflow Rate on Drying Process in a Spray Dryer Using Computational Fluid Dynamics Method

Samsudin Anis^{1,*}, Shilly Muttashillatul Urfi¹, Adhi Kusumastuti², Wim Widyo Baskoro³

¹ Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Semarang, 50229 Gunungpati, Semarang, Central Java, Indonesia

² Faculty of Engineering, Universitas Negeri Semarang, 50229 Gunungpati, Semarang, Central Java, Indonesia

³ Department of Engineering and Product Development, Inka Multi Solusi, 63151 Madiun, East Java, Indonesia

ARTICLE INFO

Article history:

Received 13 November 2021

Received in revised form 5 March 2022

Accepted 8 March 2022

Available online 10 April 2022

Keywords:

Spray dryer; drying chamber; CFD; temperature; airflow rate

ABSTRACT

Spray drying is a method of producing a dry powder from liquid or slurry by rapid drying with a hot air. The spray drying process consists of atomizing the liquid into fine droplet, contacting the hot air with the droplets, drying droplets by the formation of solid particle agglomerates and separating solid products. This research was carried out to analyse the drying process parameter of spray dryer to reach the fluid drying temperature of around 100°C in which the drying process is assumed to be successful. Simulation using computational fluid dynamics method was performed. Air and water were employed as the working two phase fluids injected into the drying chamber at various inlet temperatures and airflow velocities. RNG k- ϵ and k- ω were used to solve drying process phenomenon in the drying chamber. In this work, the inlet air temperatures and the airflow velocities were varied from 175°C to 225°C and 1.5 m/s to 3 m/s, respectively. The results showed that the higher inlet air temperature, the greater the outlet temperature. In addition, the outlet temperature also increased with the increase of airflow velocity. Under the condition investigated, the optimal parameter was found at inlet temperature of 225°C and airflow velocity of 3 m/s. This condition is expected in avoiding the failure of spray drying process.

1. Introduction

A spray dryer is a device that uses a hot gas to rapidly dry a liquid or slurry into a dry powder. The spray drying process entails atomizing the liquid feed into a fine droplet spray, interacting the droplets with drying air, drying the droplets by forming solid particle agglomerates, and separating the solid product from the flue gas. The liquid is disintegrated into droplets as a result of atomization, which then dry into particles. Droplet size varies from a few to several hundred microns, depending on the type of atomizer and the physicochemical qualities of the liquid. The droplets/particles collide as a result of air recirculation and frequent changes in flow trajectory. There are three types of

*Corresponding author.

E-mail address: samsudin_anis@mail.unnes.ac.id

collisions during the spray drying process, namely droplet-droplet collisions, droplet-particle collisions, and particle-particle collisions [1].

The inlet temperature and airflow rate of the drying chamber have a major impact on the drying process [2]. High inlet temperatures and hot air flow rates are required to transfer energy and speed up the drying process, so that the feed liquid is completely dry. Higher drying temperatures in combination with smaller droplets provide positive effects on the powder product such as higher yields, lower moisture content, less dense particles, etc. [3]. The water content of about 4% could be achieved in the spray drying process with an inlet temperature of 120°C and a drying gas flow rate of 4 kg/h [3]. Similar phenomenon of the high drying rate at high inlet temperature has also been reported in other studies [4-6]. According to the high inlet temperature, a large heat gradient exists between the atomized feed and the drying air, causing rapid evaporation of water in the sprayed material, resulting in a powder with a low moisture content [7].

Evaluation of the multi-phase drying process in the spray dryer has also been carried out through a simulation process using computational fluid dynamics (CFD) method [8,9]. This is because CFD can consider multi-phase transport phenomena. Okada *et al.*, [10] estimate the evaporation rate of water in spray drying process using a computational fluid dynamics technique, which included determining the CFD model to describe the spray dryer system and boundary conditions. Effects of drying conditions including drying air velocity and temperature, and liquid feed rate on evaporation rate have been studied. They found that the temperature difference and droplet diameter effectively affect evaporation rate. Several CFD models have also been examined for drying rate in a spray dryer and found that there is no significantly different in drying rate prediction [11]. However, the main problem in dryer operation is flow stability, where a very unstable flow can cause wall deposition of partially dry product adhering to the wall, which over time has the potential to form crust [12]. For this reason, setting the drying parameters through modelling and simulation is very necessary so that this problem can be avoided when applied in the field.

Based on the above literatures, although studies on the performance analysis of spray dryer have been done and studied at various parameters, however, different geometries and sizes will provide specific drying phenomena. This encourages the important of research on analysis of drying process of spray dryer under various inlet temperature and airflow rate with the aid of computational fluid dynamics method. Therefore, the purpose of this study was to examine the drying process of spray dryer in order to achieve optimum operating parameter in which the drying process to be successful. In the present research, a 3D geometry model was designed and used for optimizing the inlet temperature and airflow rate to obtain outlet temperature of around 100°C using CFD.

2. Methodology

Computational fluid dynamics, or CFD, is an analytical method that involves computer-based simulations of fluid flow, heat transport, and related phenomena such as chemical reactions [13]. CFD codes are based on a numerical algorithm for resolving fluid flow issues. All commercial CFD packages have a sophisticated user interface for entering issue parameters and checking the results, allowing quick access to their resolving case.

2.1 Model Characteristics

The Navier-Stokes equation is used to calculate this method, which is done numerically. The Navier-Stokes equation [14] explains the link between momentum, mass, and energy conservation in fluids. Ansys Mesh software is used in the geometry and meshing process, which entails

constructing geometric models, nodes and elements for fluid flow, and identifying boundary conditions.

Figure 1 shows the schematic representation of the drying chamber. The designed spray dryer used in this work has a hot air intake line on the side of the drying chamber cover, while the nozzle is on the top side of the drying chamber cover. Then the fluid outlet line is at the bottom of the drying chamber, which is opposite to the hot air intake. This spray dryer has a drying chamber height of 1.17 m and a diameter of 0.24 m. The drying capacity of the spray dryer was 600 mL/h. The liquid feed rate is adjusted to the drying capacity of the spray dryer. In this study, the liquid feed velocity was set at 0.031 m/s.

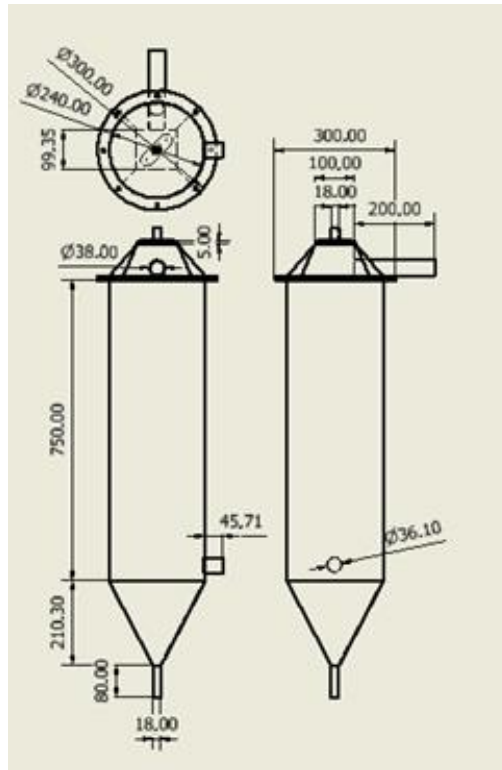


Fig. 1. Schematic representation of drying chamber (all dimensions are in mm)

2.2 Boundary Conditions

Temperatures of 175°C, 200°C, and 225°C were applied at the drying chamber's entrance, as well as airflow rates of 1.5 m/s and 3 m/s. The unstable state geometry is currently being created with Ansys Fluent software. The identification of the fluid flow and the concentration of the contour will be the first steps. The problem is identified using the spray dryer that is already in place. After that, a literature review was conducted to obtain crucial information regarding spray dryer theory, particularly the inlet temperature and dryer air flow rate throughout the drying process. The next step is to build the 3D spray dryer design process. The drying chamber inlet velocity is determined by calculating the output discharge and input discharge in the drying chamber. The simulation is started with the input geometry that was previously created on the workbench, and then the specified parameters are entered including inlet temperature, airflow rate, viscosity of the liquid feed, density of liquid feed, and pressure drop. The convective heat transfer coefficient of 294.13 W/m².°C was also determined.

Inside the drying chamber, the Reynolds number is 548285, and the Prandtl number is 1.20. The flow inside the drying chamber is laminar, while the flow in the centre is turbulent. The investigation was carried out using the k-epsilon turbulence model. The k-epsilon model accurately predicts turbulent flow at the centre and laminar flow at the drying chamber wall, which were otherwise incorrectly anticipated by RNG k- ϵ , k- ω models, as several authors have argued [13-14]. Table 1 shows the properties of the employed fluids. The simulation process requires the properties of the working fluids such as specific heat of the fluid, viscosity, density, enthalpy, etc.

Table 1
Properties of the working fluids

Properties	Air	Water
Density (kg/m ³)	1.225	998.2
Specific Heat (J/kg.K)	1006.43	4182
Thermal Conductivity (W/m.K)	0.0242	0.6
Viscosity (kg/m.s)	1.7894×10^{-5}	0.001003
Molecular Weight (kg/kgmol)	28.966	18.0152
Standard State Enthalpy (J/kgmol)	0	-2.858×10^8

2.3 Model Validation

It is required to raise the level of detail in order to approach the actual condition in order to create the highest value. Meshing detail has been increased to the highest level. The more nodes and elements that are produced, the more detailed calculations are performed on the impact. This, however, may make the computational process heavier and take longer. Figure 2 depicts the meshing process outcome and the result is tabulated in Table 2.

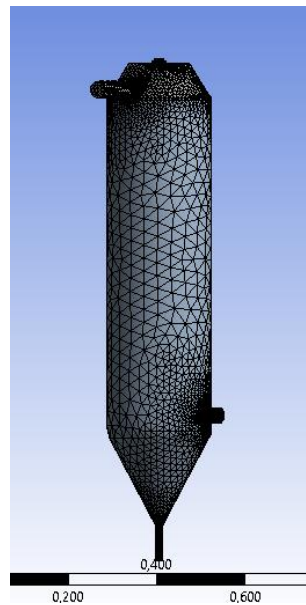


Fig. 2. Meshing result

Table 2
 Meshing result specifications

Parameter	Value
Element size (m)	1.5×10^{-2}
Bounding box diagonal (m)	1.1089
Average surface area (m ²)	7.9×10^{-2}
Min edge length (m)	5.3×10^{-2}
Mesh metric	Orthogonal quality
Standard deviation	0.13514
Number of nodes	38239
Number of elements	105891
Total volume (m ³)	3.6×10^{-2}
Inlet surface area (m ²)	4.4×10^{-2}
Outlet surface area (m ²)	9.8×10^{-4}

It is necessary to re-check the design geometry if there is a meshing failure or an error in one of the design components. One of the causes of meshing process failure is meshing value that is not set according to the design (can use program-controlled) or discrepancies during the tool assembly process. After the meshing procedure is complete, the next step to be done is numerical analysis.

The current study was validated using the results of a previous experimental study [15] that covers the impact of air inlet temperature on the spray dryer's temperature distribution published. The input temperature value utilized for comparison is 225°C with a pressure of 2 kg_f/cm² and a hot air velocity of 11.3 m/s [15].

Figure 3 depicts the temperature distribution in the drying chamber components equipped with the flow direction, as displayed by the validation model. The heat gun produces a temperature of 220°C in the drying chamber component, while the lowest temperature is 28°C. The temperature distribution was examined in order to determine an appropriate outlet temperature for drying process in the drying chamber.

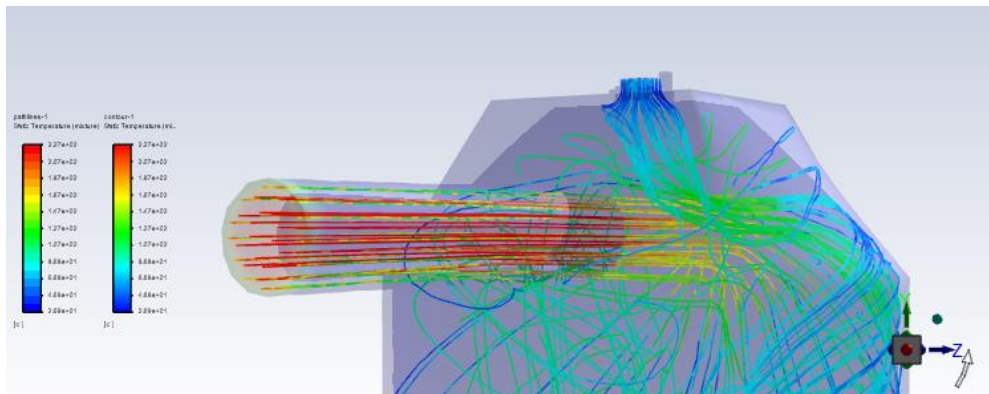


Fig. 3. Drying chamber model validation

The maximum outlet temperature in the current research model was found to be 125°C that is slightly higher compared to the result of a previous study of 123.6°C [15]. Nevertheless, the difference is very small, about 1% only, which means that the simulation result has good accuracy in predicting the temperature distribution in the drying chamber.

3. CFD Computations Results

The simulated model's scope is assumed to be free of gravity's influence. The spray dryer's intake flow rate of 600 ml/h is adjusted by the compressed air for liquid feed atomization, resulting in an inlet feed velocity of 0.031 m/s. The viscous model is k-epsilon, realizable, and provides an improved wall treatment. k-epsilon is used to determine the turbulent flow that needs to be calculated.

The ability to include pressure gradients or heat effects in computations is made possible by a realistic and upgraded wall treatment. The volume of fluid multiphase model was utilized, with air as the first phase and liquid feed as the second phase. CFD simulations utilizing Ansys Fluent 2020 R1 software were used to test the drying chamber components on a spray dryer, assuming the entire body is a fluid.

The defined model's CFD analysis is carried out by plotting various airflow velocity and temperature contours, and then analysing the results. It should be noted that the simulation results of heat transfer on the component of the spray dryer or drying chamber use liquid feed in the form of water a temperature of 70°C. Figure 4 depicts the temperature distribution within the drying chamber in the form streamline. The red colour represents a maximum temperature of 225°C at the inlet of the drying chamber, with an inlet velocity of the hot air of 3 m/s. The temperature of this process dropped from 225°C to 125°C at the outlet line.

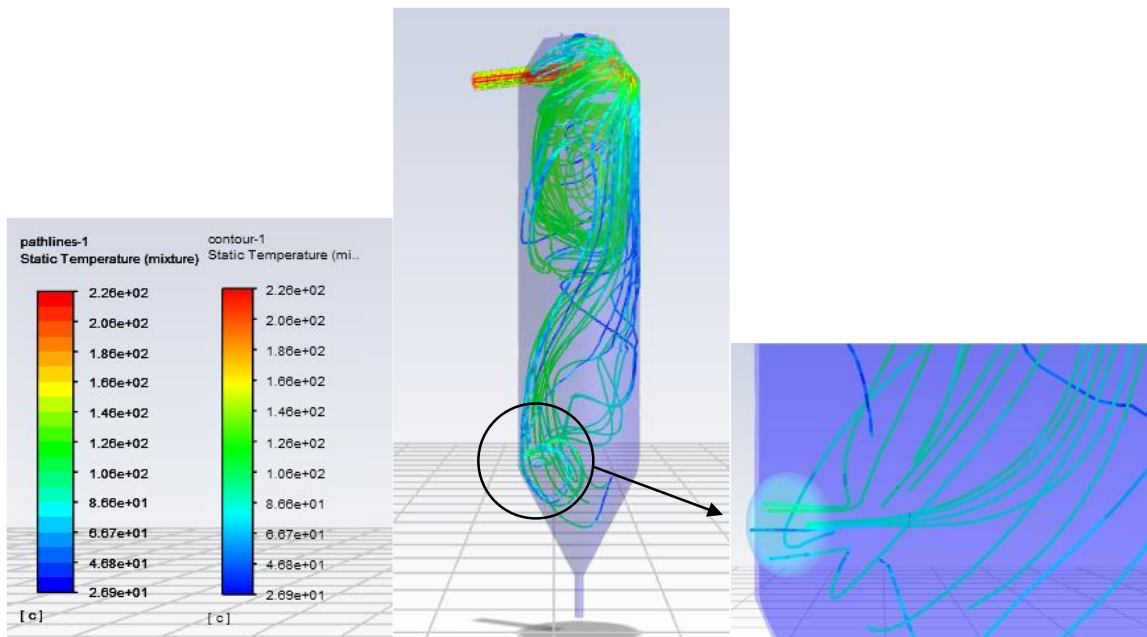


Fig. 4. Temperature streamline plot for inlet air temperature of 225 °C and air velocity of 3 m/s

Simulation results of other parameter indicated similar trend where temperature decreased along the chamber. Data showed that for inlet air temperatures of 200°C and 175°C with an airflow velocity of 3 m/s produced maximum outlet temperatures of 114°C and 101°C, respectively. Whereas for inlet air temperatures of 225°C, 200°C, and 175°C with an airflow velocity of 1.5 m/s produced relatively lower maximum outlet temperatures of 106°C, 96.3°C, and 86.2°C, respectively.

Figure 5 shows the fluid velocity streamline in the drying chamber for 3 m/s inlet airflow velocity and air temperature of 225°C. The airflow velocity of CFD simulation changes along the drying chamber. The maximum airflow velocity was found to be 4.17 m/s at the outlet line, whereas the smallest airflow velocity of 0.4 m/s occurred in the tube body. At the outlet side, the airflow velocity increased from 2.5 m/s to 4.17 m/s due to contraction. In the meantime, for inlet airflow velocity of

1.5 m/s, the maximum airflow velocity was found to be 2.07 m/s at the outlet line. In this outlet region, the airflow velocity increased from 1.24 m/s to 2.07 m/s. The increase in airflow velocity at the outlet occurred due to the addition of fluid velocity from the dried liquid feed which undergoes a phase change from liquid to gas. It is also related to the mass balance of the drying process.

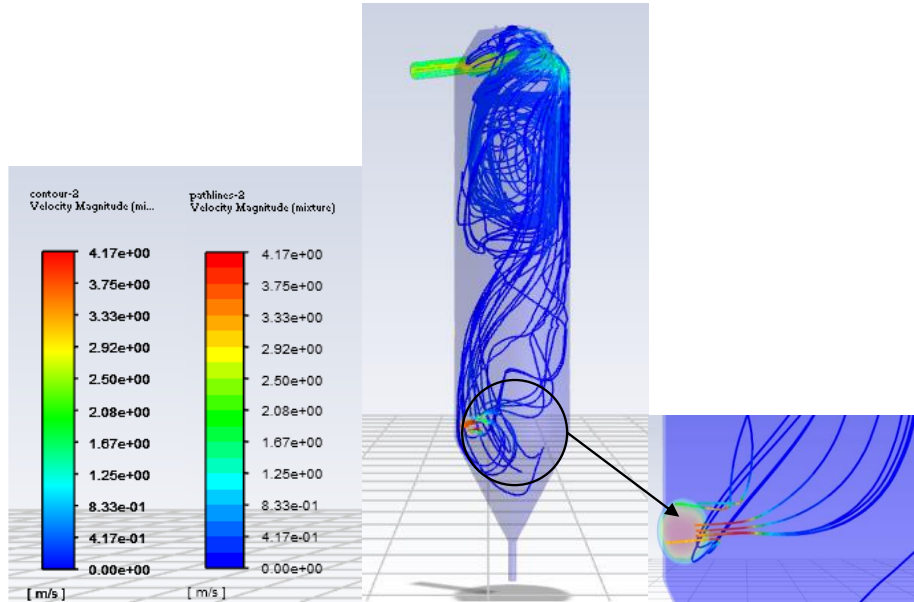


Fig. 5. Fluid velocity streamline plot for inlet air velocity of 3 m/s and air temperature of 225°C

Based on the simulation results, the influence of the inlet temperature and velocity of the air on the drying process indicated by the outlet temperature of the fluid can be observed in Figure 6. The figure shows that the smaller the inlet air temperature, the smaller the outlet temperature. This showed that the liquid feed drying process will be faster when using a high inlet air temperature. The simulation results are in accordance with the research conducted by Aguiar *et al.*, [2] regarding the effect of temperature on water content in which the higher the drying temperature, the lower the water content of the material. This phenomenon is also in line with the results of several previous studies [3-6]. However, the inlet air temperature needs to be adjusted in such a way to get good drying results. Too high inlet drying temperatures can cause undesirable powder properties [4]. In this case, if the liquid feed is a mixture of water and particles, then the outlet fluid temperature is expected to be at the evaporation temperature of the water.

Figure 6 also illustrates that higher inlet velocity of the hot air tended to generate higher outlet temperature. It can be seen that there was significant increase of outlet fluid temperature by an average of about 18% when the inlet air velocity changed from 1.5 m/s to 3 m/s. The results showed that the inlet air velocity of 3 m/s only required an inlet air temperature of 175°C to get an outlet temperature close to 100°C. In the meantime, the required inlet air temperature was about 1.3-fold higher when using a lower inlet air velocity of 1.5 m/s. This is because the volume of hot air that enters the drying chamber is more at high air velocities. This condition will increase the diffusion of hot air into the mist of the feed, which in turn increases the amount of water that can be evaporated. Thus, the high drying air velocity speeds up the feed drying process. In addition, the high speed of the drying air helps to reduce the deposition of particles on the walls, thereby increasing product yield [3,6].

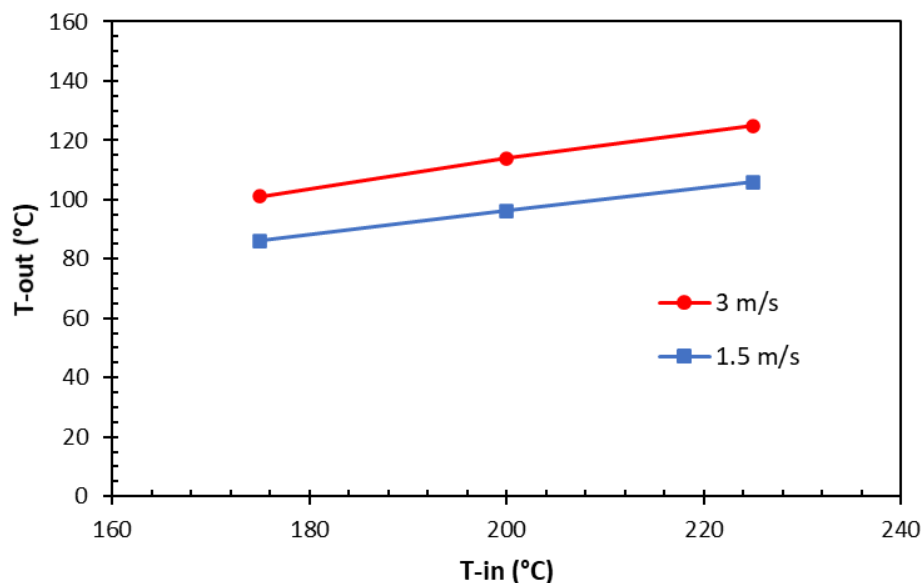


Fig. 6. Fluid inlet-outlet temperature relation for inlet air velocity of 1.5 m/s and 3 m/s

4. Conclusions

A CFD simulation of drying process on a spray dryer has been carried out as the effect of inlet air temperature and air velocity. The results of this simulation provided useful information regarding the operating conditions of the spray dryer which determine the success of the drying process. Depending on the geometry and dimensions of the spray dryer, the inlet air temperature and the inlet air velocity must be adjusted to obtain the best drying operating conditions. The result also showed that a high inlet air temperature with a certain inlet air velocity accelerates the drying process. Under the condition investigated, inlet temperature of 225°C and airflow velocity of 3 m/s were sufficient to ensure a proper drying process.

Acknowledgement

This work was funded by the research grant from Ministry of Research and Technology of the Republic Indonesia/National Research and Innovation Agency No. 58.25.3/UN37/PPK.6.8/2021.

References

- [1] Jaskulski, M., P. Wawrzyniak, and I. Zbiciński. "CFD simulations of droplet and particle agglomeration in an industrial counter-current spray dryer." *Advanced Powder Technology* 29, no. 7 (2018): 1724-1733. <https://doi.org/10.1016/j.appt.2018.04.007>
- [2] Aguiar M. C. S., M. F. d. G. F. da Silva, J. B. Fernandes, and M. R. Forim. "Evaluation of the microencapsulation of orange essential oil in biopolymers by using a spray-drying process." *Scientific Reports* 10, no. 11799 (2020): 1-11. <https://doi.org/10.1038/s41598-020-68823-4>
- [3] Dantas D., M. A. Pasquali, M. Cavalcanti-Mata, M. E. Duarte, and H. M. Lisboa. "Influence of spray drying conditions on the properties of avocado powder drink." *Food Chemistry* 266 (2018): 284-291. <https://doi.org/10.1016/j.foodchem.2018.06.016>
- [4] Himmetagaoglu A. B., and Z. Erbay. "Effects of spray drying process conditions on the quality properties of microencapsulated cream powder." *International Dairy Journal* 88 (2019): 60-70. <https://doi.org/10.1016/j.idairyj.2018.08.004>
- [5] McDonagh A. F., and L. Tajber. "The control of paracetamol particle size and surface morphology through crystallisation in a spray dryer." *Advanced Powder Technology* 31, no. 1 (2020): 287-299. <https://doi.org/10.1016/j.appt.2019.10.021>

- [6] Saha D., S. K. Nanda, and D. N. Yadav. "Optimization of spray drying process parameters for production of groundnut milk powder." *Powder Technology* 355 (2019): 417-424. <https://doi.org/10.1016/j.powtec.2019.07.066>
- [7] Golman, B., and W. Julklang. "Analysis of Heat Recovery from Spray dryer by Recirculation of Exhaust Air." *Energy Conversion and Management* 88 (2014): 641-649. <https://doi.org/10.1016/j.enconman.2014.09.012>
- [8] Ramachandran R. P., M. Akbarzadeh, J. Paliwal, and S. Cenkowski. "Computational Fluid Dynamics in Drying Process Modelling—a Technical Review." *Food and Bioprocess Technology* 11 (2018): 271–292. <https://doi.org/10.1007/s11947-017-2040-y>
- [9] Malekjani N., and S. M. Jafari. "Simulation of food drying processes by Computational Fluid Dynamics (CFD); recent advances and approaches." *Trends in Food Science & Technology* 78 (2018): 206-223. <https://doi.org/10.1016/j.tifs.2018.06.006>
- [10] Okada S., S. Ohsaki, H. Nakamura, and S. Watano. "Estimation of evaporation rate of water droplet group in spray drying process." *Chemical Engineering Science* 227 (2020): 115938-1 – 115938-11. <https://doi.org/10.1016/j.ces.2020.115938>
- [11] Woo, M. W., W. R. W. Daud, A. S. Mujumdar, Z-H. Wu, M. Z. M. Talib, and S. M. Tasirin. "CFD Evaluation of Droplet Drying Models in a Spray dryer Fitted with a Rotary Atomizer." *Drying Technology* 26, no. 10 (2008): 1180-1198. <https://doi.org/10.1080/07373930802306953>
- [12] Fletcher D. F., B. Guo, D. J. E. Harvie, T. A. G. Langrish, J. J. Nijdam, and J. Williams. "What is important in the simulation of spray dryer performance and how do current CFD models perform?" *Applied Mathematical Modelling* 30, no. 11 (2006): 1281-1292. <https://doi.org/10.1016/j.apm.2006.03.006>
- [13] Ganvir, R. B., P. Walke., and V. Kriplani. "Heat transfer characteristics in nanofluid-A review." *Renewable and Sustainable Energy Reviews* 75 (2017): 451–460. <https://doi.org/10.1016/j.rser.2016.11.010>
- [14] Jubaer, H., J. Xiao., D. Chen., C. Selomulya., and M. Woo. "Identification of regions in a spray dryer susceptible to forced agglomeration by CFD simulations." *Powder Technology* 346 (2019): 23-37. <https://doi.org/10.1016/j.powtec.2019.01.088>
- [15] Mardiyawan, A. R. Pengaruh Temperatur Inlet Udara Dan Tekanan Udara Nozzle Terhadap Distribusi Temperatur dan Efisiensi Termal Spray Dryer. *Diss unnes 2020*.