Available online www.jsaer.com

***Journal of Scientific and Engineering Research*, 2016, 3(5):72-76**

ISSN: 2394-2630
CODEN(USA): JSERBR

Research Article

Optimization of Developed Multipurpose food dryer using ANSYS

Sheidu Sumaila Onimisi[1], Victor AdekunleAdetoro[2], Nixon Chika Nosike[3], Agbadua Segun[4], Igwedi Doris Chibuzor[5]

National Engineering Design Development Institute, PMB 5082. Nnewi, Anambra State, Nigeria

Abstract The design was done in Solidworks while the analysis and simulation was performed using ANSYS FLUENT. ANSYS FLUENT is a Computational Fluid Dynamic (CFD) software in which flow fields and other physics are calculated in detail for various engineering applications. The simulation was done to analyze temperature distribution, pressure, air flow and kinetic turbulence in the system at various temperature ranging from 40, 50 and 60^{o}C. At 40, 50 and 60 ^{o}C the average temperature distribution in the system are 33.2365 ^{o}C, 42.8425 ^{o}C and 52.7850 ^{o}C. The fan speed was kept constant at 26m/s throughout the analysis, the average air velocity within the cabin remained 1.96517 m/s while exhaust air velocity was also constant at 48.7444 m/s. The maximum air pressure in the system was found to be 6453.32 Pa.

Keywords CFD, Tray Dryer, Temperature, Pressure, Kinetic Turbulence

Introduction

Computational fluid dynamics (CFD) uses powerful computers and applied mathematics to model fluid flow situations. CFD is the science of predicting; Fluid flow, Heat transfer, Mass transfer, Chemical reactions.CFD uses numerical methods and algorithms to solve and analyze problems involving fluid flows. As a developing science, CFD has received extensive attention throughout the international community since the advent of the digital computer. CFD has become an integral part of the engineering design and analysis environment of many companies because of its ability to predict the performance of new designs or processes before they are ever manufactured or implemented [1]. Researchers, equipment designers and process engineers are increasingly using CFD to analyze the flow and performance of process equipment.

Drying is a complex process because both heat and mass transfer occur at the same time. The aim of this process is not only the removal of moisture from food material, but also acquiring a quality product. On the other hand, drying needs huge amounts of energy, thus a development in any kind of drying equipment would be advantageous for reducing energy costs and quality losses of food. In this sense, a growing technology – Computational Fluid Dynamics (CFD) – is able to be used for mathematical modeling of drying phenomena and designing and optimizing of a dryer in a reliable way. Süfer Ö, in 2014 argued that CFD applications will highlight complicated flow behaviors in dryers and make it easier for optimization [2].

Cristiana B. M. et al, demonstrated that CFD can be used as a tool to predict the hydrodynamic as well as the heat- and mass-transfer mechanisms occurring in the drying units. It can also be used to better understand and design the drying equipment with less cost and effort than laboratory testing [3].

In a similar research, Suhaimi M., et al in 2013 used CFD to predict drying uniformity of a new designed of the commercial tray dryer for agricultural product. The temperature and velocity profile, streamline and velocity on each tray were analyzed to study the uniformity of the drying [4].

This work investigates the effect of the operating parameters on the quality of products and the efficiency of the system. Air velocity in the cabin, temperature distribution and static pressure distribution were studied at various air inlet temperature.

Shahab A. et al, in 2011 in his independent studies of tray dryer for fruits using CFD concluded that operating parameters are highly controlling the dryer efficiency, whereas increase in the air temperature highly reduced the energy required and enhance the dryer efficiency. These findings are not far from the reality in drying experiments [5].

Application of tray dryer is widely used in agricultural drying because of its simple design and capability to dry products at high volume. However, the greatest drawback of the tray dryer is uneven drying because of poor airflow distribution in the drying chamber. Implementing the proper design of a tray dryer system may eliminate or reduce non-uniformity of drying and increases dryer efficiency. Computational fluid dynamics simulation is a very useful tool in the optimization of the drying chamber configuration by predicting the airflow distribution and the temperature profile throughout the drying chamber [6].

There are several designs of tray dryer system with CFD. To implement proper design of tray dryer there is a need to optimize variables that influence drying operation such as drying chamber configuration, air velocity and air temperature.

Marian V. et al, in 2014 was of the view that with progressing computing power, it is conceivable that CFD will continue to provide explanations for more fluid flow, heat and mass transfer phenomena, leading to better equipment design and process control for the food industry [7].

The studies present the design and analysis of tray dryer system. The design was done using Solid works while the analysis and simulation was performed using ANSYS FLUENT. ANSYS FLUENT is Computational Fluid Dynamic (CFD) software in which flow fields and other physics are calculated in details for various engineering applications. The analysis was done to analyze heat transfer and temperature distribution, pressure, airflow and turbulence.

Brief Description of the Dryer

The multipurpose dryer is a batch convective dryer using heated air as the drying medium and operating in batch wise mode. The dryer is a direct heat tray type achieving heat exchange through direct contact between the hot air and the material to be dried. The dryer is designed to operate under adiabatic condition. The gas/solid contacting pattern in the dryer is parallel flow in which the direction of air flow is parallel to the surface of the yam bed. Contacting is primarily at the interface between the air phase and yam bed, with possibly some penetration of air into the voids among the sliced yam near the surface. The yam bed is in a static (stationary) condition. The multipurpose dryer has metal frame structure which holds the four trays in position. The design is so flexible that it can be operated by one person.

The dryer dimensions are 1.0 × 0.6 × 1.2 (m). It is fabricated from metal sheet and the inside is made of composite material with insulator made of fiber glass of thickness 20mm.

Methodology

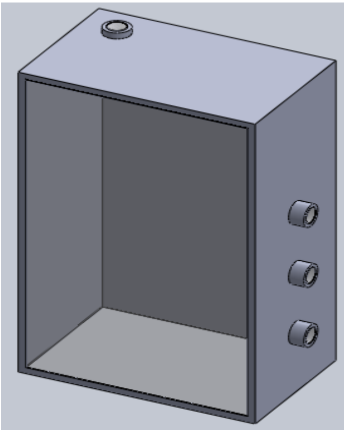

Figure 1: Solid works model of the dryer exposing the inside without the trays

The Cabin dryer was modelled in Solidworks and imported to ANSYS 14.0 for the analysis. ANSYS Fluent was used for the Computational Fluid Dynamic (CFD) analysis. Inlet temperature of 40 ^{o}C, 50 ^{o}C and 60 ^{o}C of air at inlet and constant air velocity of 26m/s was used, the minimum and maximum air temperature distribution in the cabin were monitored, also investigated are the exhaust temperature and air velocity at various air inlet temperature.

Discussion of Results

Table1: CFD results

Drying temperature	Min. tempt in cabin	Max. tempt in cabin	Avg. Tempt. In cabin	Fan speed	Air velocity in cabin	Exhaust air velocity	Exhaust air Tempt
40	25.1900	41.2830	33.2365	26.00	1.96517	48.7444	39.205
50	34.4180	51.2670	42.8425	26.00	1.96517	48.7444	
60	44.2010	61.3690	52.7850	26.00	1.96517	48.7444	59.267

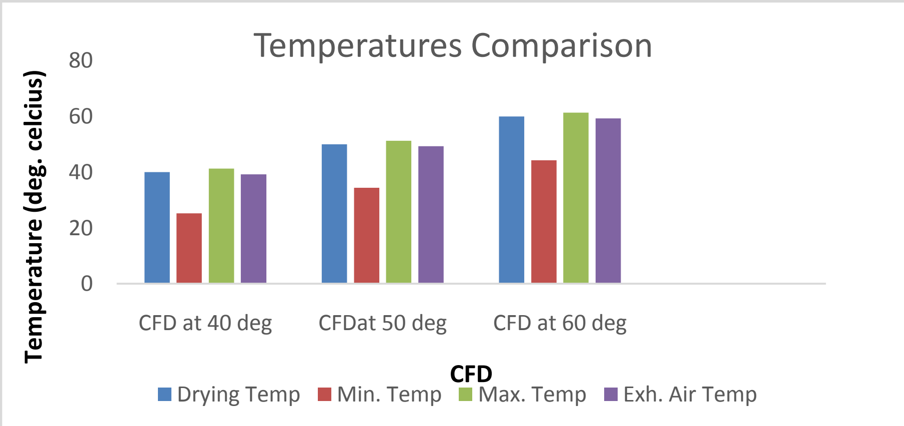

Figure1: Pictorial View of the Temperatures

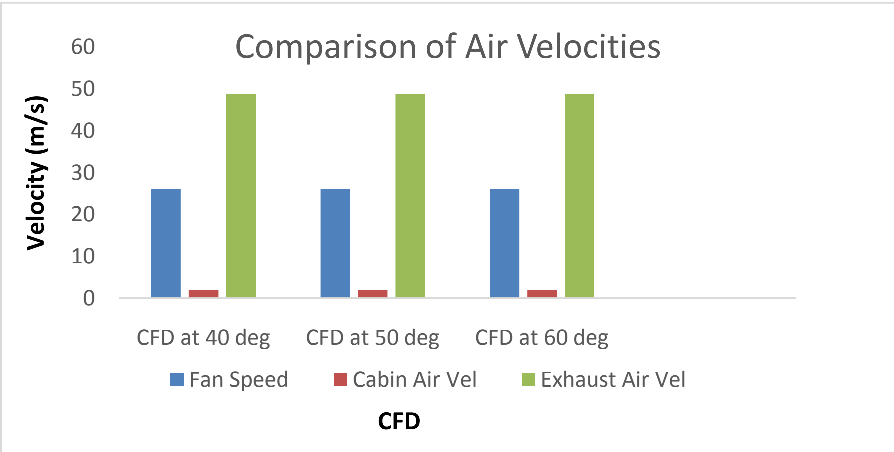

Figure2: Pictorial View of the Air Velocities

The computational fluid dynamics shows that when the inlet temperature of air is 60 ^{o}C and the control is set at 60 ^{o}C the minimum and maximum temperature in the system are 44.2010 ^{o}C and 61.3690 ^{o}C with an average of 52.7850 ^{o}C. At 50 ^{o}C inlet air and set temperature the minimum and maximum obtainable are 34.4180 ^{o}C and 51.267 ^{o}C with the average air temperature distribution of 42.8425 ^{o}C in the system.

While at 40 ^{o}C inlet and set temperature the minimum temperature in the system is 25.19 ^{o}C and a maximum of 41.283 ^{o}C with an air temperature distribution in the system of 33.2365 ^{o}C. In this investigation the fan speed was kept constant at 26 m^3/s, the velocity within the cabin remained 1.96517 m/s and the air velocity at 48.744 m/s. The exhaust air temperature increases correspondingly as shown in the figure 1.

Table1.2: CFD results

Drying temperature	Pressure Min. (Pa)	Pressure max. (Pa)	Air velocity in cabin (m/s)	Min. Turbulence Kinetic Energy (J/kg)	Max. Turbulence Kinetic Energy (J/kg)
40	6430.72	6453.32	1.96517	1.20784	42.2994
50	6430.72	6453.32	1.96517	1.20784	42.2994
60	6430.72	6453.72	1.96517	1.20784	42.2994

The lower the magnitude of the turbulence and kinetic energy the more stable the flow. The maximum value obtained during simulation is 42.2994 J/kg which indicates that the flow is in stable condition. The minimum and maximum pressure in the system are 6430.72 Pa and 6453.32 Pa.

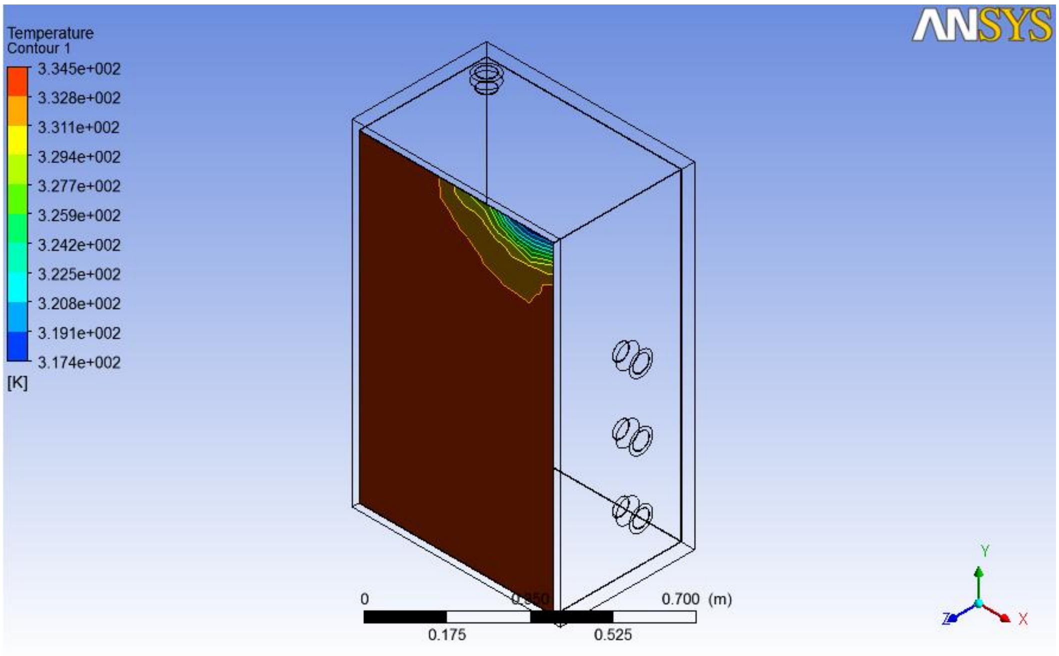

Figure 3: Temperature Profile in Symmetry of the Cabin

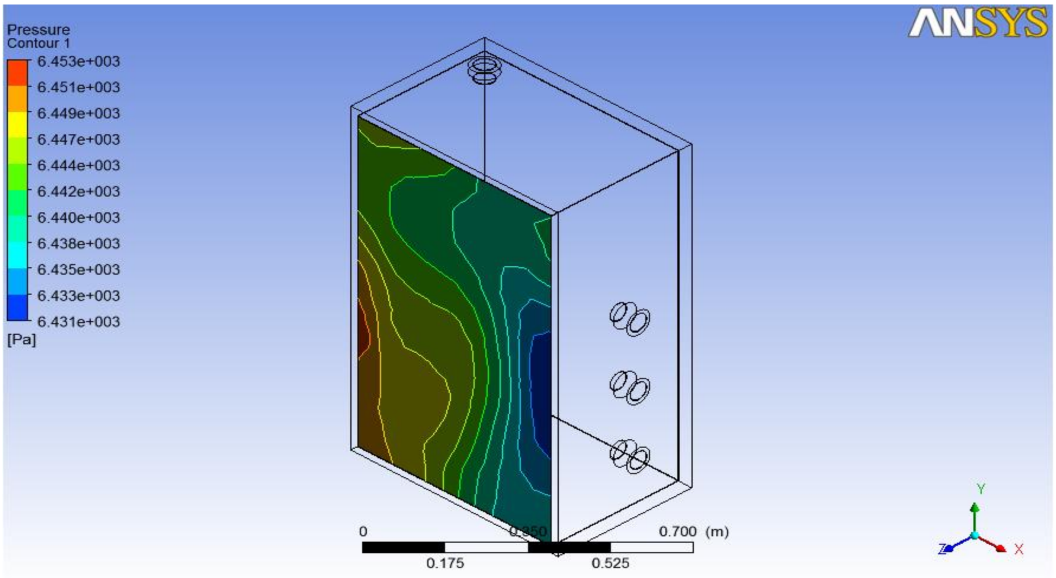

Figure 4: Pressure Profile in the Cabin

The temperature is evenly distributed in the cabin due to the strategic location of the fans. The fans run continuously even if the automatic control switch off the air heater due to the temperature set. The pressure distribution in the system is shown by figure 4 above. The maximum air pressure in the system is 6453.32 Pa.

Conclusion

The computational fluid dynamic (CFD) analysis for the tray dryer system is presented. The air distribution was even and adequate at the design fan speed as shown by the CFD which also translated to even and adequate temperature distribution within the system. The maximum pressure in the system is 6453.32 Pa. The pressure in the system is only affected by the air velocity and temperature changes have no effect on the pressure build up on the system. The temperature of the outlet air is slightly lower than the cabin inside temperature.

References

[1]. Bin Xia and Da-Wen Sun, 2002. Applications of computational fluid dynamics (CFD) in the food industry: a review. 2002 Elsevier Science B.V. PII: S0168-1699(01)00177-6.

[2]. Süfer O., (Nov, 2014). An Emergent Technology for Modelling Drying: Computational Fluid Dynamics (Cfd). 2nd International Congress on Food Technology; November 05-07, 2014

[3]. Cristiana B. M. et al (March, 2012). Simulation of The Airflow Inside A Hybrid Dryer. International Journal of Recent Research and Applied Studies. Vol 10 Issue 3.

[4]. Suhaimi M., et al 2013. The Prediction of Drying Uniformity in Tray Dryer System using CFD Simulation. International Journal of Machine Learning and Computing, Vol. 3, No. 5, October 2013.

[5]. Shahab A. et al, 2011. Fruit drying process: Analysis, modeling and simulation. Scientific Research and Essays Vol. 6(23), pp. 4915-4924, 16 October, 2011.

[6]. Misha S, et al, 2013. Review on the Application of a Tray Dryer System for Agricultural Products. World Applied Sciences Journal 22 (3): 424-433, 2013. ISSN 1818-4952.

[7]. Marian V. et al, 2014. Prediction of Air Flow and Temperature Profiles inside Convective Solar Dryer. Bulletin UASVM Food Science and Technology 71(2) / 2014. Electronic ISSN 2344-5300.