

Michael Mayokun Odewole *et. al.* (2017). *Computational fluid dynamics (CFD) simulation of hot air flow pattern in cabinet drying of osmo-pretreated green bell pepper*. 12(1)36-45.

Computational fluid dynamics (CFD) simulation of hot air flow pattern in cabinet drying of osmo-pretreated green bell pepper

Michael Mayokun Odewole¹, Musliu Olushola Sunmonu², Samuel Kehinde Oyeniyi^{3*} and Oluwatosin Adedamola Adesoye⁴

^{1,2}Department of Food Engineering, Faculty of Engineering and Technology, University of Ilorin, Ilorin, Nigeria.

³Department of Agricultural and Environmental Engineering, Faculty of Technology, University of Ibadan, Ibadan, Nigeria.

⁴Department of Mechanical Engineering, Faculty of Engineering and Technology, University of Ilorin, Ilorin, Nigeria.

Abstract

Computational Fluid Dynamics simulation of air flow distribution, air velocity and pressure field pattern as it will affect moisture transient in a cabinet tray dryer is performed using SolidWorks Flow Simulation (SWFS) 2014 SP 4.0 program. The model used for the drying process in this experiment was designed with Solid Works 2014 SP4.0 while the simulation was performed using SWFS 2014 program using the standard turbulence Intensity-Length model in steady-state regime. The effects of the model were used to analyze the flow parameters and it was observed that fluid density vary along the distance of travel. The average air temperature gotten in the process of the Computational Fluid Dynamics (CFD) simulation was in agreement with the experimental preset temperature for drying green bell pepper with an error of 0.02 %. The dryer model describes the transfer process and flow field pattern of air in the drying chamber and predicts the instantaneous temperature, air velocity, pressure field pattern at any location of the dryer with emphasis on the drying trays. The R^2 values of drying parameters as they vary with distance of travel along the trays were presented in order to know the relationship of these parameters with the dryer setup.

Keywords: CFD, SolidWorks, SWFS, drying, airflow, drying temperature

Email: oyeniyisk@gmail.com.

Received: 2016/09/24

Accepted: 2017/02/07

DOI: <http://dx.doi.org/10.4314/njtr.v12i1.7>

Introduction

The trend of increasing productivities in agriculture and food availability is as a result of rapid development or advancement in drying technologies which has reduced the rate of deterioration of agricultural produce. Drying was defined as the removal of moisture from a material to a predetermined level. It was also reported that a good drying process should take into consideration three important things viz: drying temperature, drying methods and the characteristics of the materials to be dried (Mujumdar, 2007). Odewole and Olaniyan, (2015) reported that, drying can be classified as a simultaneous heat-and-mass transfer process that can be classified basically into three methods viz: traditional, improved and industrial (modern) methods. These methods came into being as a result of advancement in drying technology from the use of the sun to the use of sophisticated mechanical equipment like various forms of powered dryers of which cabinet tray dryer is one. These advancement has led to different numerical, analytical and computational simulation of drying process but few on moisture transient and drying rate which is a function of air flow or air velocity during drying operation. These two parameters are difficult to measure analytically due to its complexity and required measuring devices

that are needed to be placed at various points within the drying chamber.

Amanlou and Zomorodian (2010) reviewed the application of CFD in food equipment design with emphasis laid on rechecking the simulation results by experiment because CFD use many approximate models as well as few assumptions. According to Dimitrios *et. al* (2013) it was concluded that one of the efficient ways to evaluate the quality of airflow inside the dryer is by applying computational fluid dynamics (CFD) to predict the flow pattern. Ryu *et. al* (2013) used CFD technique to calculate the characteristics of air flow inside a dryer unit and the results gotten were compared with the outcome of the experiment to validate the potential of CFD simulation. CFD potential was strengthened by the fact that it can be adopted in order to minimize the costly experiment because users can generate visual results in similitude to the real life situation (Lone *et. al.*, (2005); Kaushal and Sharma. (2013)). Numerical simulation of some drying parameters inside a drying chamber were studied by Margaritis and Ghiaus, (2006) and a set of measurements was obtained experimentally above a single tray to validate the standard k- ϵ model. The validation of the measured data and the simulation results through CFD showed that the standard k- ϵ model is the most adequate

turbulence model. In the light of this validation, this present study advanced it by varying drying parameters over three trays and used standard turbulence Intensity–Length model (2 % and 0.006 m) to validate the variation of drying parameters along the drying trays. In a nut shell, understanding flow characteristics in the dryer is of the utmost important since it is one of the major functions that are responsible for adequate drying process.

Tray dryer system is characterized by the following basic features: drying chamber, heating source and a fan to pass air over the product to evaporate the moisture into vapour condition. The air enters the bottom of the chamber below the trays, rises through the trays, and exits through the ducting system or opening on the top of the chamber. In view of this arrangement, the drying rate are being affected because the products located on the bottom trays are more dried than those on the upper trays, which are subjected to decreased temperature and air velocity. Also, it was stated that tray dryers are the most widely used dryers for various drying applications because it is cost effective and simple in terms of its design. But the major challenge in tray dryer is in heat distribution which leads to lower quality in final product (Misha *et. al.*, 2013). Generally, moisture removal is an essential prerequisite for long term storage of food products under normal temperature and humidity conditions (Michael and Ojha, 2003). The rate at which moisture is being removed from materials can be hastened by the drying rate which is a strong function of the air temperature and velocity of the drying air. Though it has been reported that these two parameters are difficult to measure during the drying process because several sensors would be needed to be placed at various directions of air flow and locations within the drying chamber (Pinaga *et. al.*, 1987).

In view of the aforementioned challenges, Computational fluid dynamics (CFD) stands to ameliorate this great challenge in determining flow characteristics in the dryer since it has the capability to use numerical methods and algorithms to solve and analyze problems that involve fluid flows and heat transfer. In CFD analysis, computers are used to perform the mathematical calculations required to simulate the interaction of liquids and gases with

surfaces defined by boundary conditions. CFD is used in predicting what will happen quantitatively when fluids flow, often with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction, mechanical movement, stresses in and displacement of immersed or surrounding solids during drying of food products. The main significance of implementing CFD analysis is to solve complex flow behavior which is not possible with ordinary calculus. Also, measuring the drying parameters in the drying chamber is expensive, difficult, and time consuming because sensors and data loggers have to be installed in several positions, particularly in large scale dryers. Therefore the objectives of this research were to use CFD to simulate some measurable parameters during drying operation in a cabinet dryer and standard I-L model for the turbulence effect as regards the drying parameters.

Materials and methods

Description of the Dryer

A cabinet tray dryer was designed and constructed as shown in Figure 1-2. The dryer has three major sections; these are the drying chamber, the cylindrical heating chamber which houses the heater and the blower (axial fan) and the temperature controlling unit. The drying chamber has a rectangular cross-section, and it is double walled (25 mm thick) insulated with fibre glass to minimize heat loss through the chamber walls to the surroundings. The drying chamber walls were made of 1.5 mm gauge galvanized steel sheet. The external dimension is 600 mm (width) x 600 mm (length) x 750 mm (height), while the internal dimension is 550 mm (width) x 550 mm (length) x 700 mm (height). It has three trays which were stacked with a distance of 150 mm to one another in a vertical arrangement to ensure optimum exposed to drying air and to improve airflow distribution throughout the drying chamber. Each tray has an area of 530 mm x 530 mm and was made from 20 mm square pipe. It has heating chamber of a cylindrical configuration of length 150 mm and diameter 200 mm which houses the fan and the heating element. Heat is supplied by an electrical heating coil of 1800 W and hot air is blown from the heater housing to the drying chamber by the aid of an axial fan as shown in Figure 1 below. The dryer also consists of a

digital temperature regulator for maintaining a drying temperature of 55 °C. Also, provision was made for air outlet in order to disallow moisture condensation at the top of the drying chamber; a duct of 80 mm by 80 mm was made at the top of the drying chamber. Green bell peppers were loaded on the drying trays, the cabinet is closed and heated air of about 55 °C is blown across.

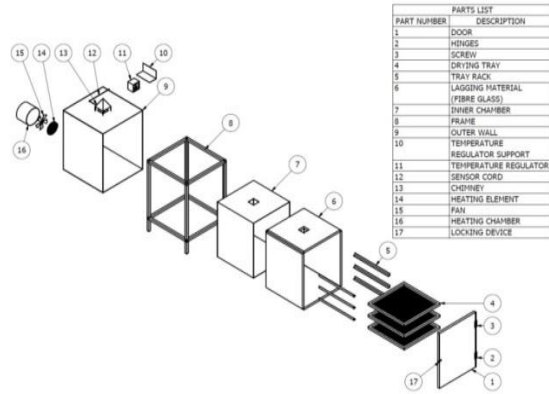


Figure 1: Assembly drawing of the dryer

Experimental Materials and Procedures

The green bell pepper was procured from the central market in Ilorin, Kwara State. The total weight of the sample in all trays was approximately 1.5 kg. The pepper was sliced to thickness of 3 mm and dried without the seeds. The initial moisture content was determined by oven-drying method to approximately 85 % wet basis. All samples passed through osmotic dehydration pretreatment process in hypertonic salt (NaCl) solutions under the following conditions: osmotic solution concentration (5% (w/w), 10% (w/w), 15% (w/w), 20% (w/w) and 25% (w/w)) and osmotic process duration of 60 min, 90 min, 120 min, 150 min and 180 min. Osmotic dehydration was done as means of improving the sensory (colour, flavour, odour and texture) and nutritional qualities (vitamins, minerals and others) of the products after drying. All samples were arranged in the dryer after the osmotic dehydration pretreatment process.

Computational Fluid Dynamics Modeling

Basic Governing Equations for Modeling Fluid Flow

The basis governing equations for CFD simulation modeling are the Navier-Stokes equations which were derived from the basic mass conservation and continuity equations applied to properties of fluids. In order to derive the equations of fluid motion, the

continuity equations (which dictates conditions under which things are conserved) was first derived, then equation for conservation of mass and momentum, and finally combine the energy equation. These equations were iteratively solved using CFD solvers internally.

The three equations that describe Navier-Stokes equations are as follows:

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial U_i}{\partial x_i} = 0 \quad (1)$$

Momentum Equation

$$\underbrace{\rho \frac{\partial U_j}{\partial t}}_I + \underbrace{\rho U_i \frac{\partial U_j}{\partial x_i}}_{II} = - \underbrace{\frac{\partial P}{\partial x_j}}_{III} - \underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\frac{\rho g_j}{V}}_V \quad (2)$$

Energy Equation:

$$\underbrace{\rho c_\mu \frac{\partial T}{\partial t}}_I + \underbrace{\rho c_\mu U_i \frac{\partial T}{\partial x_i}}_{II} = - \underbrace{P \frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda \frac{\partial^2 T}{\partial x_i^2}}_{IV} - \underbrace{\tau_{ij} \frac{\partial U_j}{\partial x_i}}_V \quad (3)$$

I: Local change with time

II: Momentum convection

III: Surface force

IV: Molecular-dependent momentum exchange (diffusion)

V: Mass force

Where:

$$\tau_{ij} = -\mu \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \frac{2}{3} \delta_{ij} \mu \frac{\partial U_k}{\partial x_k} \quad (4)$$

ρ is the fluid density (kg/m³), U is the fluid velocity vector (m/s), τ_{ij} is the viscous stress

tensor, P is pressure (Pa), $\frac{\partial U_j}{\partial x_i}$ is the body

forces (N), and T is time (s), (i, j, k) depicts the three main coordinates which the mass force depends on, c_μ is the fourth-order tensor representing the constant of proportionality, λ is the viscosity tensor, μ the dynamic viscosity (Ns/m³) and δ_{ij} is the Kronecker delta.

Finite Volume Method approach was used in this simulation to discretize Navier-Stokes equation, because it can automatically control the conservation of mass, momentum and energy in the whole domain and this is done by finding the integral form of the Navier-Stokes equation over a control volume and by applying Gauss Theory as presented below

$$\begin{aligned} \frac{\partial(\rho\phi)}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho U_i \phi - \Gamma \frac{\partial}{\partial x_i} \right) \\ = q_\phi \quad \phi \\ = \{1, U_j, T\} \end{aligned} \quad (5)$$

Integrating over the Control Volume (CV)

$$\begin{aligned} \int_V \frac{\partial}{\partial t} \phi dV \\ = \int_S \phi \cdot n_i dS \end{aligned} \quad (6)$$

Integral Form of Navier-Stokes Equation

$$\begin{aligned} \underbrace{\int_V \frac{\partial(\rho\phi)}{\partial t} dV}_I + \underbrace{\int_S \left(\rho U_i \phi - \Gamma \frac{\partial}{\partial x_i} \right) \cdot n_i dS}_{II} \\ = \underbrace{\int_V q_\phi dV}_{III} \end{aligned} \quad (7)$$

Where:

I: Local change with time in CV

II: Flux over the CV Surface

III: Source in CV

Geometry of the Cabinet dryer

The 3D view of the considered cabinet dryer is shown in Figure 2. The drying unit system consists of a drying chamber inside which there are three trays stacked vertically, axial fan and a heating element. Thereafter, the geometry was meshed in order to cope-up with the thermal and velocity boundary layer formation



Figure 2: Geometry of the model in SolidWorks 2014 SP4.0

The numerical finite volume method used in SolidWorks Flow Simulation (SWFS) 2014 SP 4.0 was used to solve Equation (1) to (3). Unstructured refined 3D grid mesh was used to build a numerical model. The simulation was conducted under steady state condition because the study focused on the pattern of the air stream in the drying chamber.

Boundary conditions

The boundary conditions and computational fluid domain are shown in Figure 3 and 4 respectively. Some parameters and properties which were used during analysis are as follows:

1. Air properties: this is presented in table 1 below.

Table 1: Physical parameters and properties

No	Physical Parameters	Remarks
1	Fluid	Air
2	Density of air	1.127 kg/m ³
3	Specific heat (Cp)	4182 J/kg.K
4	Thermal conductivity	0.0242 W/m.K
5	Viscosity	1.7894e-05 kg/m.s
6	Velocity magnitude (Inlet)	3.02 m/s
7	Temperature (Inlet)	323 K
8	Gauge Pressure	101325 Pa
9	Velocity magnitude (Outlet)	2.91 m/s

2. Porous media: The trays were assumed to be porous with 20% porosity.
3. Wall: The heat transfer coefficient of the chamber wall is 4 W/m²K and environmental conditions were defined. The environmental temperature was assumed to be 35.5 °C.
4. Heating element: The heat transfer rate was set at 1800 W and the temperature was controlled to 55 °C

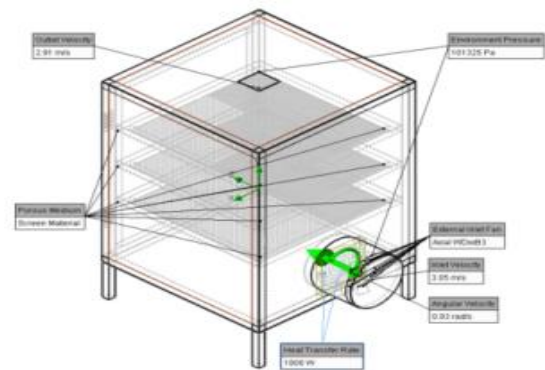


Figure 3: Drying chamber layout and boundary conditions.

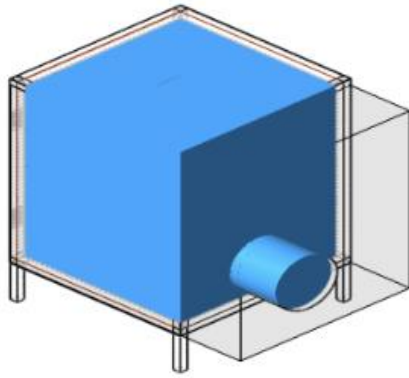


Figure 4: Geometry of the model showing the fluid domain

Study of hot air flow characteristic inside cabinet dryer by using CFD method

For this simulation, the standard turbulence intensity of 2.00 % and length of 0.006 m with enhanced wall treatment were employed to simulate the temperature distribution and turbulence field. These equations were iteratively solved internally using SolidWorks Flow Simulation (SWFS) 2014 SP 4.0, which is based on Finite Volume Method (FVM). In order to obtain the correct simulated results, In the course of the solving process the total cell count, Fluid Cells, Solid Cells and Partial Cells of 370972, 148107, 84033 and 138832 respectively were gotten from discretization process and were used to analyze the simulation. Furthermore, the simulated average temperature in the drying chamber was validated by comparing with the measured temperature as presented in Table 2 with percentage error of 0.02. Based on this result it can be concluded that this model is in a good agreement with experiment and the model can be adopted to analyze the phenomena inside the dryer.

Table 2: The average predicted temperature and measured temperature in the dryer

	Experiment	CFD (Predicted)	% Error
Temperature (°C)	55	56.2	0.02

Results and discussion

In drying application, the temperature, velocity and humidity of drying air condition gave significant effect to the drying process. In this simulation study, some major parameters of the drying air were analyzed especially above

the trays since the materials to be dried will be placed on the drying trays.

CFD post processing of temperature distribution in the dryer

The uniformity of hot air flow distribution in the drying chamber can be assessed from the Front plane cut plot of the temperature profile since it cuts across all the trays. Though several planes were studied but the plane (Figure 5) closest to the rear end of the drying chamber presented a poor zone compare to the cut plot at the middle (Figure 6) of the drying chamber in terms of hot air distribution due to its location and the effect of the fan. The hot air temperature from inlet is 55 °C. It was found that for porous product the average temperature at the three trays were in the range of 48.7 to 63.7 °C.

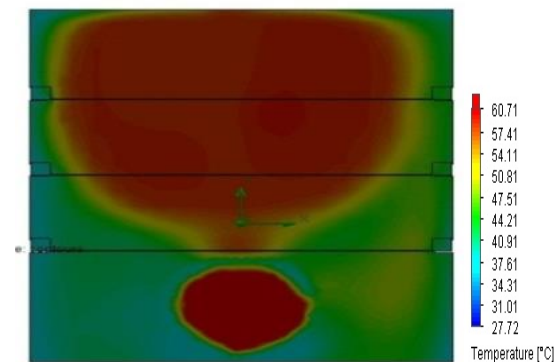


Figure 5: Temperature Cut Plot at the rear end of the drying chamber showing poor zone

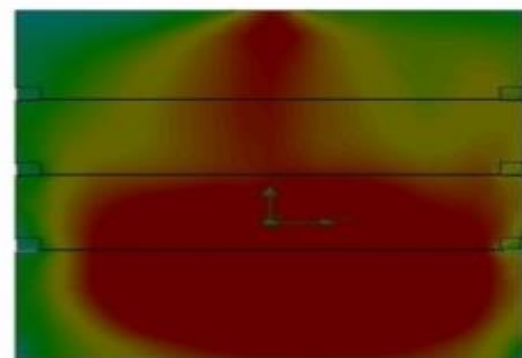
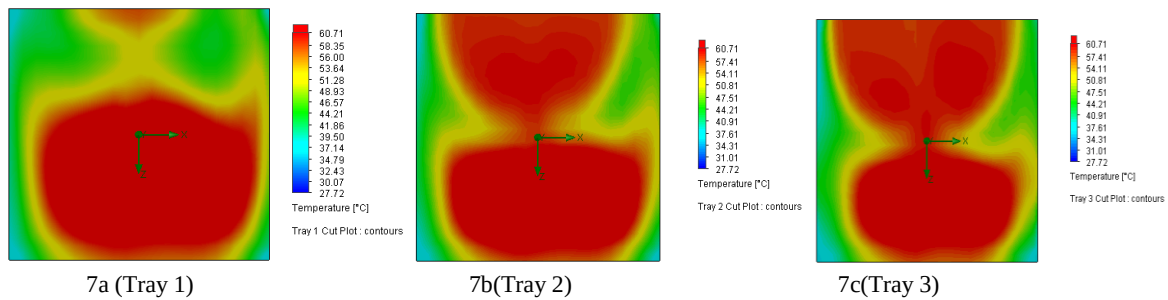


Figure 6: Temperature Cut Plot at the middle of the drying chamber showing relatively uniform distribution

Temperature contours of each tray in the dryer are presented in Figure 7(a)-7(c). From this result, it can be said that the center of the drying chamber receives appreciable hot air needed for drying operation and this is a function of fan location and direction of flow which has a great impact in temperature distribution within the dryer.



Figures 7(a)-7(c): CFD post processing of temperature distribution on the trays (Trays 1-3 from the base)

Also, figure 8 show that the temperature contours grew from the heater but not well distributed at the rare of the drying chamber, since the air enters the dryer at ambient temperature being heated by the electric heater. The average temperature in the drying chamber is in accordance with experimental data. The experimental value was 55 °C. In the simulation, a temperature of 56.2 °C was obtained which shows a perfect representation of the phenomenon in the dryer. It is of high necessity to note that the walls are well insulated in the experimental setup in order to minimize heat transfer through the dryer walls, and were simulated considering adiabatic wall in the numerical model. It can be seen that the distribution of temperatures inside the dryer in the vertical plane is almost homogeneous, except at the fan location (air inlet) where lower temperatures are obtained, of about 28.3 °C, as well as, the highest temperature gradients, of about 73.7 °C at the heater region. Overall, the mean temperature gradient

between the dryer inlet and outlet regions is approximately equal to 56 °C, which is a reasonable value for drying.

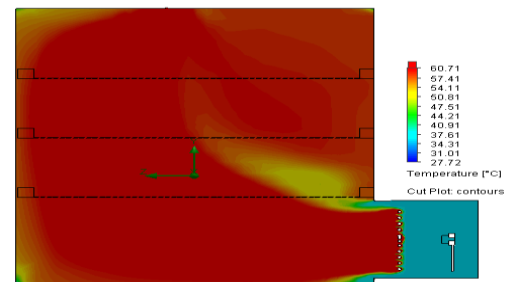


Figure 8: Temperature field inside the dryer (middle section).

CFD post processing of velocity fields in the dryer.

Figure 9 presents the velocity field of the airflow on the drying trays. The velocity increases towards the exit section i.e. the duct. This behaviour was expected, since the flow cross-section decreases in this direction.

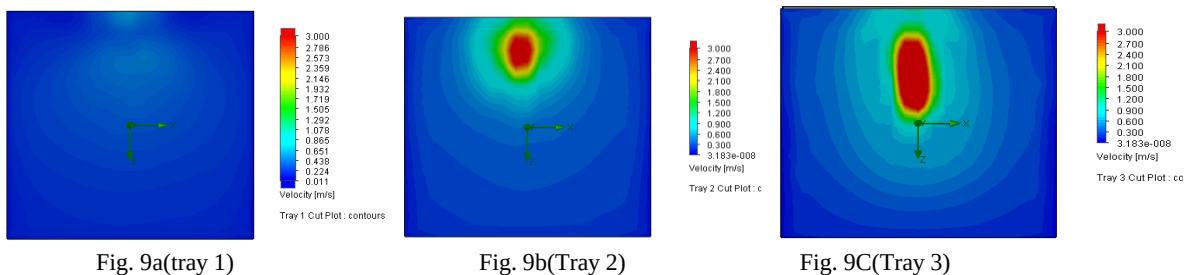


Fig. 9a(tray 1)

Fig. 9b(Tray 2)

Fig. 9C(Tray 3)

Figures 9(a)-9(c): CFD post processing of velocity field on the trays (1-3 from the base)

In Figure 10 below is displayed the velocity field inside a middle vertical plane (side view) in the dryer. The region with the fan and the duct region depict the highest velocity and this is as a result of the openings at both ends. Also, it was observed that airflow velocity towards the door produce the lowest velocity profile and this is in agreement with the Newton second law of motion. This flow pattern can be explained by the contour cut plots as displayed in Figure 10. As observed in this figure (and previously explained in Figure

8 as regards temperature distribution) the hottest air coming from the electric heater forms a recirculation region in the drying chamber vertical direction, flowing first near the drying chamber backward wall, and then turning around the other walls (top and frontal walls) of the drying chamber, finally flowing again near the backward wall to the exit region located at the top of the drying chamber wall but it was also observed that air flows from the surrounding to the drying chamber from the opening (duct) provided. This flow pattern is

in agreement with the cross-sectional temperature distribution plotted in Figure 8.

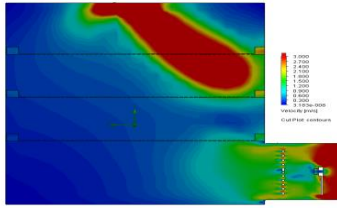


Figure 10: Velocity field inside a middle vertical plane (side view) in the dryer.

CFD post processing of pressure fields in the dryer.

As shown in figures 11 and 12, static pressure of the drying system is concentrated on the heating region because of the fan and maximum pressure recorded from the simulation is 137797.70 Pa. This is as a result of induced air being introduced by the fan curves.

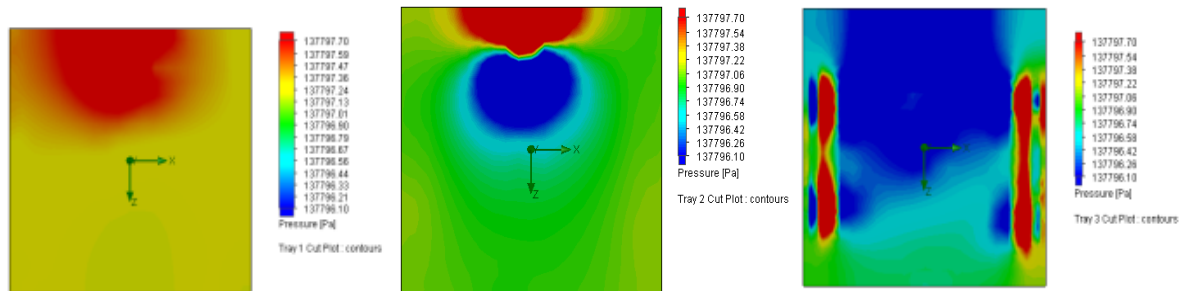


Figure 11: CFD post processing of pressure field on the trays (1-3 from the base)

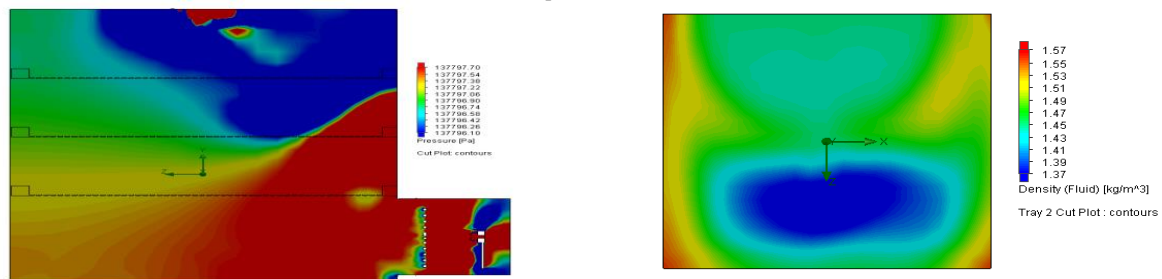


Figure 12: Pressure field inside a middle vertical plane (side view) in the dryer.

CFD post processing of density of fluid fields in the dryer

It was observed that the fluid density in the drying chamber is not constant as reported using k- ϵ models but I-L model was able to predict the pattern of fluid density in the dryer better because it did not assume fluid density to be constant. Figure 13 presented the cut plots for fluid density in which tray 1 displayed high fluid density in the range of 1.3-1.57 kg/m³ and this is as a result of the turbulence level. Though in fluid mechanics, density of incompressible fluid are said to be constant. With this result, it can be said that air is a not an incompressible fluid but a compressible fluid.

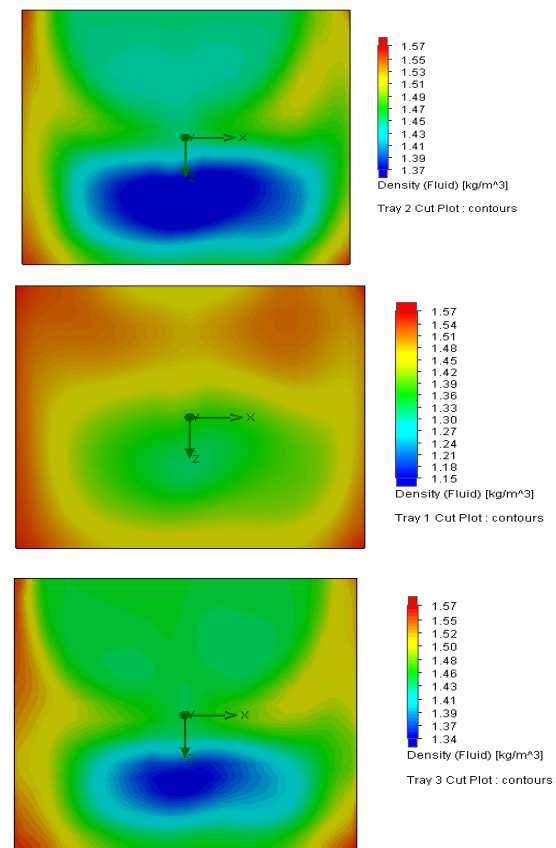


Figure 13: CFD post processing of density of fluid field on the trays (1-3 from the base)

CFD post processing of Flow Trajectories of Temperature, Air Velocity and Pressure fields in the dryer

The flow trajectories of air temperature, velocity and pressure are presented in Figure 14 below to predict the flow characteristics within the drying chamber right from the inlet to the outlet (duct). The drawn flow trajectories were produced with 0.000001 m thickness of lines with arrows and 10000 points were used to produce a perfect visualized result of the flow trajectories. It was observed that the temperature is concentrated at the center of the dryer, maximum velocity were gotten at the inlet region due to the presence of the fan and also the maximum pressure were gotten at the first layer (Tray 1).

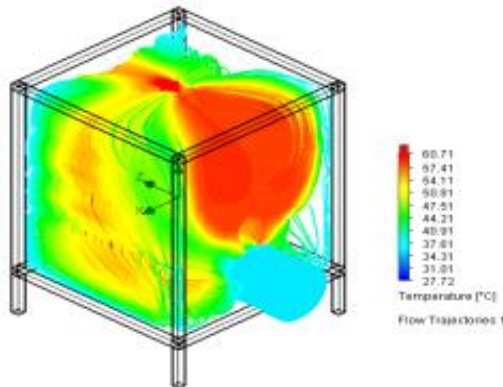


Figure 14(a): Flow Trajectories of temperature

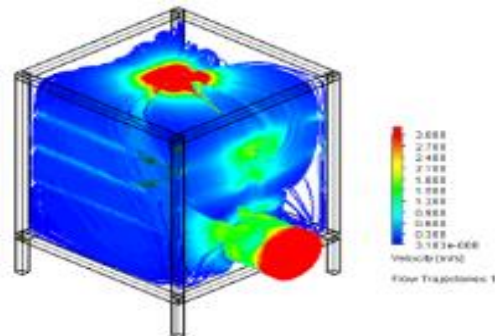


Figure 14(b): Flow Trajectories of Velocity

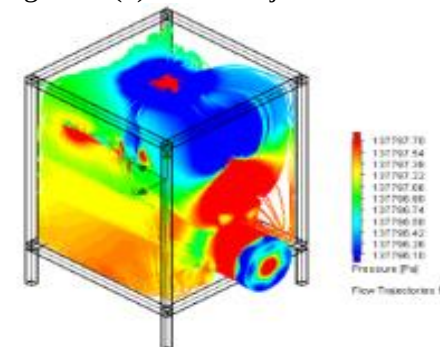


Figure 14(c): Flow Trajectories of Pressure

Figure 14: Flow Trajectories of Temperature, Air velocity and Pressure in the drying chamber

Variation of drying parameters in respect to the arrangement of the trays

Drying parameters such as temperature, air velocity and pressure were varied across each of the trays in order to predict the part that received appreciable heat most during the drying operation.

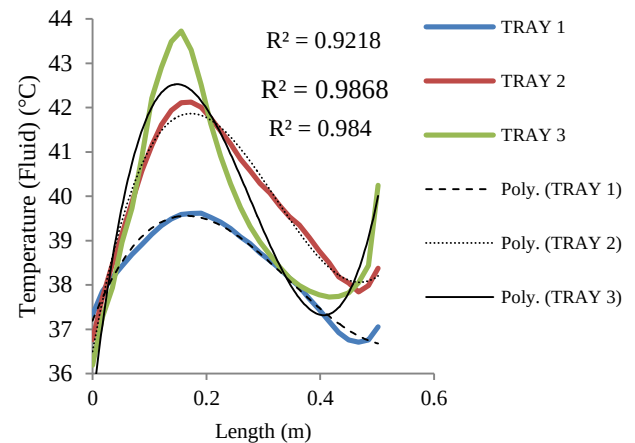


Figure 15: Temperature Profile at Various points on the tray with respect to length.

This plot (Figure 15) shows the temperature profile of the fluid flow in relationship to the arrangement of the trays in the drying chamber. It can be seen that temperature at Tray 2 has the highest R^2 value of 0.9868 compare to Tray 1 and 3 of R^2 values of 0.984 and 0.9218 respectively. This depicts that the temperature of the fluid vary with the distance of travel best along tray 2 and the percentage of hot air it contributed to at this level is greater than others. Also, it was observed that there are peaks period at the distance between 0.15-0.2 m along the trays and the temperature later drops towards the end of the trays due to the collision of a boundary (dryer door) with lower temperature which is in agreement with the law of conservation of energy. A decreasing temperature profile was also obtained by Turner and Jolly, (1991) and Zhang and Mujumdar (1991) for microwave convective drying and Golestani *et al.* (2013) for convective drying simulations due to the fact that in simulation, heat loss due to conduction is considered negligible. However, they reported these results without any interpretation of this event. To investigate this observation further, the temperature of the fluid in relationship with length of travel across the trays were plotted in a single graph as shown in Fig. 15. The surge in temperature

level was observed at a distance 0.8 m from the rare of the dryer wall on tray 3 because of the flow trajectory of the hot air as a result of the upwind direction of the flow. This phenomenon is important in food drying where an increase in temperature can cause quality degradation. If the arrangements of food materials to be dried on the tray are in good agreement with the hot air flow, the incessant of uneven dried product in cabinet dryer will be solved. More experimentation with continuous temperature measurement should be undertaken to further validate this phenomenon. As outlined above, temperature distribution at any instance is important because spoilage can start from regions with poor distribution of it and caking of the products can occur at the region with undesirable level. From the plot (Figure 15), it is possible to determine the effective length of the drying chamber and recirculation points which is also in agreement with the study conducted by Karim and Hawlader, (2005).

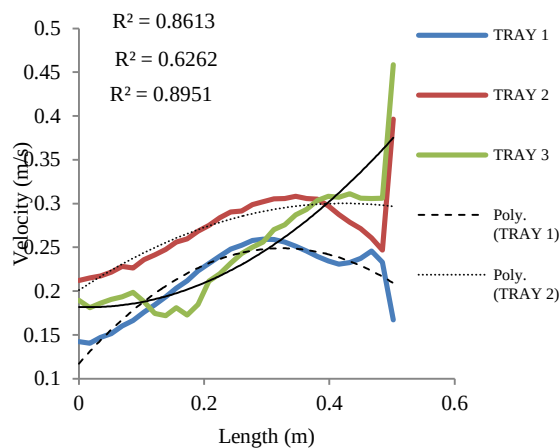


Figure 16: Velocity Profiles at Various points on the tray with respect to length

This plot (Figure 16) shows the velocity profile of the fluid flow in relationship with the arrangement of the trays. It can be seen that air velocity at Tray 1 has the highest R^2 value of 0.8951 which depicts that the velocity of the flow vary with the distance of travel best along tray 1. This has to do with the dryer geometry, it can be seen that areas with the highest maximum velocity are occurring at the ends of the trays 2 and 3 but not in agreement with tray 1 due to its location in the drying chamber.

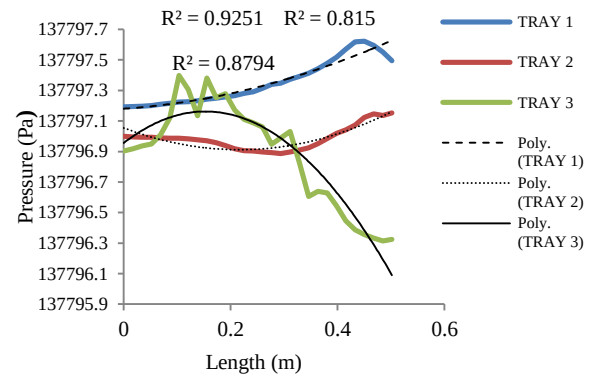


Figure 17: Pressure Profiles at Various points on the tray with respect to length

This plot (Figure 17) shows the pressure profile of the fluid flow at along the length of the trays. It can be seen that Tray 3 was characterized with the highest pressure drop towards the end of the drying tray and this is as a result of relatively undisturbed air at that level while Tray 1 pressure profile increases towards the end of the tray with the R^2 value of 0.9251 which depicts that the pressure of the flow vary with the distance of travel best along tray 1.

Conclusions

The design and analysis for the cabinet tray dryer system is presented and computational fluid dynamic (CFD) simulation was used to predict temperature distribution, velocity and pressure fields in the drying chamber by considering the product as porous media in similitude to agricultural materials. The temperature distribution, velocity field profiles and pressure were also analyzed to predict the efficiency of the Cabinet dryer. The result shows that the experimental drying temperature and simulation data were in good agreement. From the study, one can adjudged that the drying rate of the product would be significantly influenced by the air velocity, the drying air temperatures and the arrangement of the trays. It can be concluded I-L turbulence model is capable of predicting the dynamic behavior of the dryer. Also, the R^2 values of drying parameters as they vary with distance of travel along the trays were presented in order to know the relationship of these parameters with the dryer setup. The temperature at Tray 2 has the highest R^2 value of 0.9868 while air velocity and pressure at Tray 1 has the highest values of 0.8951 and 0.9251 respectively. Thus, CFD stands at the

leading edge in predicting airflow pattern and analysis of drying parameters. CFD application to dryer design can agreeably be said to be one of the most efficient and effective solutions to uneven dried products.

Acknowledgements

The authors wished to appreciate department of Food and Bioprocess Engineering, University of Ilorin for their support in releasing their workshop for the fabrication of the dryer and laboratory for the drying experiment.

References

- Amanlou, Y. and A. Zomorodian, (2010). Applying CFD for designing a new fruit cabinet dryer, *Journal of Food Engineering*, 101: 2010: 8-15.
- Dimitrios, A.T., Achilleas, V.B., Alexandros, P.V., Dionysios, G.T., Dimitrios, A.C., Andronikos, E.F. and P. M. Dionissios (2013). Design, Construction and Performance Evaluation of a New Laboratory Convective Dryer. 5th International Conference on Experiments Process System Modeling Simulation and Optimization 5th IC-EpsMsO Athens, 3-6 July, 2013
- Golestani, R., Raisi, A. and A. Aroujalian (2013). Mathematical modeling on air drying of apples considering shrinkage and variable diffusion coefficient. *Drying Technology* 2013: 31(1): 40–51.
- Karim, M.A., and M.N.A Hawlader,. (2005). Mathematical modeling and experimental investigation of tropical fruits drying. *International Journal of Heat and Mass Transfer* 2005: 48, 4914-4925
- Kaushal, P. and H. K. Sharma. (2013). Concept of Computational Fluid Dynamics (CFD) and its Applications in Food Processing Equipment Design. *J. Food Process Technol.* 3:138. 2013: DOI:10.4172/2157-7110.1000138
- Kumar, C., Karim, A., Saha, C., Joardder, M.U.H., Brown, R. and D. Biswas (2012). Multiphysics Modelling of Convective Drying of Food Materials. *Proceedings of the Global Engineering, Science and Technology Conference*. 28-29 Dec. 2012. Dhaka, Bangladesh.
- Lone, H.M., Monika, W., Raluca, H. and R. Gilles (2005). CFD Modelling of Moisture Interactions between Air and Constructions. *Building Simulation Ninth International IBPSA Conference Montréal, Canada August 15-18, 2005*.
- Margaris, D.P. and A. G. Ghiaus (2006). Dried product quality improvement by air flow manipulation in tray dryers, *Journal of Food Engineering*, 75: 542–550.
- Mathioulakis, E., Karathanos, V.T. and V.G. Belessiotis, (1998). Simulation of air movement in a dryer by computational fluid dynamics: Application for the drying of fruits, *Journal of Food Engineering*, 36: .183–200.
- Michael A.M. and T. P Ojha. (2007). Principles of Agricultural Engineering Vol. I and Vol II Jam Brothers, New Delhi. <http://www.tnps.gov.insyllabus/dgagricultural.engineering.pdf>
- Misha, S., Mat, S., Ruslan, M. H., Sopian, K. and E. Salleh, (2013). The prediction of drying uniformity in tray dryer system using CFD simulation, *International Journal of Machine Learning and Computing*, 3(5): 419–423.
- Mujumdar, A.S. (2007). *Handbook of Industrial Drying Fourth Edition* CRC Press Taylor and Francis Group London, New York.
- Odewole, M.M. and A.M Olaniyan,. (2015). *Empirical Modeling of Drying Rate and Qualities of Red Bell Pepper*. LAP LAMBERT Academic Publishing.
- Pinaga, F., Mulet, A., Berna, A. and M. Borras, (1987). Effect of air flow rate on carrot drying, *Drying Technology*, 5(2): 245–258.
- Ryu, J.B., Jung, C.Y. and S. C. Yi, (2013). Three-dimensional simulation of humid-air dryer using computational fluid dynamics, *Journal of Industrial and Engineering Chemistry*, 19: 1092-1098
- Turner, I.W. and P. C. Jolly (1991). Combined microwave and convective drying of a porous material. *Drying Technology* : 9(5): 1209–1269.
- Zhang, W., and A. S. Mujumdar, (1992). Deformation and stress analysis of porous capillary bodies during intermittent volumetric thermal drying. *Drying Technology* 10(2): 421–443.