CFD ANALYSIS OF TEMPERATURE AND VELOCITY DISTRIBUTION IN FORCED CONVECTIVE CABINET DRYER UNIT

[1]Asif Basha A [2]Nataraj M,[3]Kannan Ramasany,
[1]PG Scholar, [2] Professor,[3]Research scholar,
[1,2]Department of Mechanical Engineering, [3]Anna University,
[1,2]Government College of Technology, Coimbatore, Tamilnadu, India.

Abstract: Cabinet dryers are the most popular drying devices for fruit drying. One of the drawbacks of this dryer can be non-uniformity in the desired end product moisture content. To overcome this problem, a new version of a cabinet dryer with the vertical divergent and convergent chamber is designed for the research. In Cabinet tray dryers, trays are used (product holders) to expose the product to the heated air in an enclosed space called cabinet. Heating may be an air current sweeping across the trays, by conduction from heated trays on which the trays lie, or by radiation from heated surfaces with air currents to sweep moisture. Most tray dryers are heated by air, which also removes the moist vapors from the product. In tray dryers, the materials are spread out quite thin shaped on trays in which the drying takes place. Air movement over the food products surface is at relatively high velocities to ensure that convective heat and mass transfer proceeds in an efficient manner. The design of cabinet drying chamber is similar to the solar-type dryer but in this case, the heat is supplied by electricity. The three-dimensional geometry of the cabinet dryer with two food product holding trays was studied theoretically using the Computational Fluid Dynamics (CFD) technique. The most appropriate geometrical sketch with acceptable uniform air flow rate and temperature distribution in the cabinet dryer was selected and fabricated. The main use of CFD analysis is to predict the temperature and velocity distribution in each tray and to make use of the CFD analysis results to do the experiment in an efficient manner and reduce the time and energy consumption for obtaining the end result.

Index Terms – Computational Fluid Dynamics (CFD).

I. INTRODUCTION

Cabinet dryers are the most favorite equipment used in farms for fruit drying. These dryers are simple in struc- ture, low in cost installation and can be employed in almost any environmental conditions. Non-uniformity in the moisture content of the end product is an inherent drawback in applying the cabinet dryer; hence producers are not usually interested in utilizing this drying system (Adams and Thompson 1985; Mathioulakis et al. 1998; Mirade 2003).

In conventional cabinet dryers, hot air is normally introduced under the first tray (bottom tray) and passes through the other trays. Therefore the drying materials on the bottom trays would receive the highest energy and might be over dried, while the materials on the other trays may not receive enough energy to be dried due to a decrease in drying air potentials. Dehydration rate illustrates a very strong relationship with drying air temperature and velocity (Mulet et al. 1987; Karathanos and Belessiotis 1997). Uniform airflow distribution inside the drying compartment is of paramount importance because it determines both the efficiency and the homogeneity of the products being dried (Mirade 2003).

Controlling all of these effective parameters experimentally is a very tedious and difficult task Although the Computational Fluid Dynamics technique (CFD) cannot replace physical experiments completely but it can significantly reduce the amount of time needed for experimental works. This valuable tool is capable of analyzing the flow pattern of the air conditioning system in a short span of time, which was previously impossible from experimental and theoretical methods (Anderson 1995; Yongson, et al. 2007). Computational fluid dynamics (CFD) have been extensively applied for predicting the air velocity and temperature profiles in drying chambers (Norton and Sun 2006).

With the development of low cost, powerful computers, and commercial software packages such as Fluent, Star-CD, and CFX in the last decade, CFD has been increasingly deployed in the food industry (Scott and Richardson 1997; Verboven, et al. 2000a, b; Xia and Sun 2002). CFD has been effectively used to study odors dispersion. CFD can consider various atmospheric phenomena and topographical conditions to study the occurrence of odors and aerosol dispersions (Hong et al. 2011).

CFD simulation of the four commercial models of corrugated cellulose evaporative cooling pads that are most widely used in Mediterranean greenhouses (A. Franco, et al. 2011). CFD to evaluate the climate distribution in an animal's body and livestock's thermal environment (Norton et al. 2010). Accuracy of prediction can be strongly improved by including some pertinent physical properties of fruits such as air flow resistance, kernel and bulk densities and porosity in the study (Fluent 6.3 User's Guide 2005).

A new cabinet dryer with a side mounted plenum chamber was designed using CFD, constructed and evaluated by Amanlou and Zomorodian 2010. Their experiments were conducted on the most appropriate sketch with acceptable uniform air flow and temperature distribution. Comparing the experimental and predicted (extracted for the CFD analysis) data revealed a very good correlation coefficient of 0.99 and 0.86 for drying air temperature and air velocity in the drying chamber, respectively.

II. LITERATURE REVIEW

Adams and Thompson et al., (1985) Cabinet dryers are the most favorite equipment used in farms for fruit drying. These dryers are simple in structure, low in cost installation and can be employed in almost any environmental conditions. Non-uniformity in the moisture content of the end product is an inherent drawback in applying the cabinet dryer; hence producers are not usually interested in utilizing this drying system.

Karathanos and Belessiotis et al., (1997) in conventional cabinet dryers, hot air is normally introduced under the first tray (bottom tray) and passes through the other trays. Therefore the drying materials on the bottom trays would receive the highest energy and might be over dried, while the materials on the other trays may not receive enough energy to be dried due to a decrease in drying air potentials. Dehydration rate illustrates a very strong relationship with drying air temperature and velocity.

Mirade (2003) et al., Uniform airflow distribution inside the drying compartment is of paramount importance because it determines both the efficiency and the homogeneity of the products being dried.

Anderson et al., (1995) Controlling all of these effective parameters experimentally is a very tedious and difficult task. Although the Computational Fluid Dynamics technique (CFD) cannot replace physical experiments completely but it can significantly reduce the amount of time needed for experimental works. This valuable tool is capable of analyzing the flow pattern of the air conditioning system in a short span of time, which was previously impossible from experimental and theoretical methods.

Norton and Sun et al., (2006) Computational fluid dynamics (CFD) have been extensively applied for predicting the air velocity and temperature profiles in drying chambers.

Delele et al., (2009) temperature uniformity assessments were the subject of many studies, in which both experimental and computational results were presented. Most of the published works considered flow and temperature distribution in devices for food as well as other branches of industry.

Navaneethakrishnan et al., (2007) conducted the CFD computations in heating ovens used in bakery shops. In their study, the oven was modeled as a 2-D steady-state natural convection heat transfer problem. In addition, the material properties of air were assumed to be constant, except for the density variations with temperature.

Mirade and Daudin et al., (2006) discussed an application of a CFD approach to predict air velocity patterns and the circulation of an exogenous gas inside a pilot cheese-ripening room using three-dimensional geometry. In their mathematical model, iso-thermal flow and turbulence equations were formulated. The authors reported a fairly close agreement between the prediction of air flow patterns within the pilot cheese-ripening chamber and measured values.

Delele et al., (2009) this technique was also successfully applied for the device improvements in the food industry. Particularly, the CFD was used to design and optimize the humidification process of cold stores.

Foster et al.., (2005) among turbulent models the standard k – epsilon model still remains an industry standard and its successful application is reported in recent literature.

III. METHODOLOGY

The present research was devoted to design a new version of the food product cabinet dryer with a vertical Divergent and Convergent design and inside it with two food products holding trays were installed. In this work CFD was employed the main objectives of this study were the following:

a) To predict the air flow and temperature distribution in each of the trays for the effective drying process.
b) CFD (Computational Fluid Dynamics) analysis is employed to track the velocity and temperature distribution effects.

It helps us to do the Experimental process in a better way and reduce the time consumption and cost for the experimental trial testing.

IV. DESIGN OF CABINET DRYER

The drying cabinet is made up of 3 mm thick mild steel and it consists of two sections, the dryer, and the diffuser section. The dryer section is a cubicle of length 0.55 m, breadth 0.55 m and height of 55 mm which houses three trays through the tray holder which is welded along the length of the drying cabinet. Drying cabinet is the most important component in the drying chamber where actual heat and mass transfer takes place. One side of the cabinet holds the door for loading and unloading of the tray into the cabinet while three sides of the dryer are completely sealed. The diffuser helps to spread the hot air from the exchanger to the drying products so that all the products will have contact with the hot air at the same time. The top of the dryer cabinet is frustum-shaped also, so as to hasten the removal of humid air from the dryer, which may result in condensation. The air which carries the moisture from the food products exits to the atmosphere through the frustum provided at the top of the cabinet. The tray holder and bodies are made of mild steel to make them strong for supporting the weight of the vegetables and tray to ensure proper aeration of the drying product.

Fig 1.The Experimental setup

4.1 Basic governing equation for designing the cabinet dryer

The governing equations solved by FLUENT and the turbulence models used for this simulation are explained. Two equation models are used for the simulations. Flow equations and energy equations are described in detail. The wall treatment methods are also discussed and how they are important for modeling the heat transfer is also described.

a) Flow Calculation

The flow is governed by the continuity equation, the energy equation, and the Navier-Stokes momentum equations. Transport of mass, energy, and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume where i,j,k = 1,2,3 correspond to the three dimensions.

b) Continuity Equation

The continuity equation describes the conservation of mass and is written as in equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_1}{\partial x_1}+\frac{\partial \rho U_2}{\partial x_2}+\frac{\partial \rho U_3}{\partial x_3}=0 \qquad (1)$$

or

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_i}{\partial x_i}=0, \quad i=1,2,3$$

Equation (1) defines the rate of increase of mass in a control volume as equal to the amount through its faces. Whereas, for constant density continuity equation is reduced to

$$\frac{\partial U_i}{\partial x_i}=0, \quad i=1,2,3$$

c) Momentum Equations (Navier-Stokes Equations)

The momentum balances, also known as Navier-Stokes equations, follow Newton's second law: The change in momentum in all directions equals the sum of forces acting in those directions. There are two different kinds of forces acting on a finite volume element, surface forces, and body forces. Surface forces include pressure and viscous forces and body forces include gravity, centrifugal and electromagnetic forces.

The momentum equation in the tensor notation for a Newtonian fluid can be written as in equation (2).

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i}+v\frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)+g_i \qquad (2)$$

The equation (2) can be written in different forms for constant density and viscosity since $v\frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) = v\frac{\partial^2 U_i}{\partial x_j \partial x_i}$ in incompressible flow. In addition to gravity, there can be further external sources that may affect the acceleration of fluid e.g. electrical and magnetic fields. Strictly it is the momentum equations that form the Navier-Stokes equations but sometimes the continuity and momentum equations together are called the Navier-Stokes equations. The Navier-Stokes equations are limited to macroscopic conditions.

The continuity equation is difficult to solve numerically. In CFD programs, the continuity equation is often combined with the momentum equation to form a Poisson equation (3). For constant density and viscosity, the new equation can be written as below.

$$\frac{\partial}{\partial x_i}\left(\frac{\partial P}{\partial x_i}\right) = -\frac{\partial}{\partial x_i}\left(\frac{\partial\left(\rho U_i U_j\right)}{\partial x_i}\right) \qquad (3)$$

This equation has more suitable numerical properties and can be solved by proper iteration methods.

d) Energy Equation

Energy is present in many forms in flow i.e. as kinetic energy due to the mass and velocity of the fluid, as thermal energy, and as chemically bounded energy. Thus the total energy can be defined as the sum of all these energies.

$$h = h_m + h_T + h_C + ø \quad (4)$$

$h_m = \frac{1}{2}\rho U_i U_j$ Kinetic energy

$h_T = \Sigma_n m_n \int_{T_{ref}}^{T} C_{p,n}\, dT$ Thermal energy

$h_C = \Sigma_n m_n h_n$ Chemical energy

$ø = g_i x_i$ Potential energy

In the above equations m_n and $C_{p,n}$ are the mass fraction and specific heat for species n. The transport equation for total energy can be written with the help of the above equations. The coupling between energy equations and momentum equations is very weak for incompressible flow, thus equations for kinetic and thermal energies can be written separately. The chemical energy is not included because there were no species of transport involved in this project.

The transport equation for kinetic energy can be written as under,

$$\frac{\partial(h_m)}{\partial t} = -U_j\frac{\partial(h_m)}{\partial x_j} + P\frac{\partial U_i}{\partial x_i} - \frac{\partial(PU_i)}{\partial x_i} - \frac{\partial}{\partial x_j}\left(T_{ij}U_i\right) - T_{ij}\frac{\partial U_i}{\partial x_j + \rho g U_i} \quad (5)$$

The last term in equation (5) is the work done by the gravity force. Similarly, a balance for heat can be formulated generally by simply adding the source terms from the kinetic energy equation.

$$\frac{\partial(\rho C_p T)}{\partial t} = -U_j\frac{\partial(\rho C_p T)}{\partial x_j} + K_{eff}\frac{\partial^2 T}{\partial x_j x_j} - P\frac{\partial U_j}{\partial x_j} + T_{kj}\frac{\partial U_k}{\partial x_j} \quad (6)$$

The term on the left side of the equation is the accumulation term. The first on the right is convection term, second is the conduction, third expansion and last is dissipation term. Here the terms in the equation for transformation between thermal and kinetic energy, i.e. expansion and dissipation occur as source terms.

e) Turbulence Modeling

Realizable k – ϵ Model

The realizable k – ϵ model differs from the standard k – ϵ model in that it features realizability constraints on the predicted stress tensor, thereby giving the name of the realizable k – ϵ model. The difference comes from the correction of the k-equation where the normal stress can become negative in the standard k – ϵ model for flows with a large strain rate. This can be seen in the normal components of the Reynolds stress tensor.

$$(u_i u_i) = \sum(u_i^2) = \frac{2}{3}k - 2v_T\frac{\partial U_i}{\partial x_j} \quad (7)$$

Note that $(u_i u_i)$ must be larger than zero by definition since it is a sum of squares. For boundary layer flows, separated flow and rotating shear flows show that the realizable k – ϵ model performs better than the standard k – ϵ model.

V. CFD MODELING AND ANALYSIS OF CABINET DRYER

a) Geometry

The overall size of the drying cabinet is made up of 3 mm thick mild steel and it consists of two sections, the dryer, and the diffuser section. The dryer section is a cubicle of length 0.55 m, breadth 0.55 m and height of 55 mm which houses two trays through the tray holder which is welded along the length of the drying cabinet. Drying cabinet is the most important component in the drying chamber where actual heat and mass transfer takes place. One side of cabinet holds the door for loading and unloading of tray into the cabinet while three sides of dryer is completely sealed. The diffuser helps to spread the hot air from the exchanger to the drying products, so that all the products will have contact with the hot air at the same time.

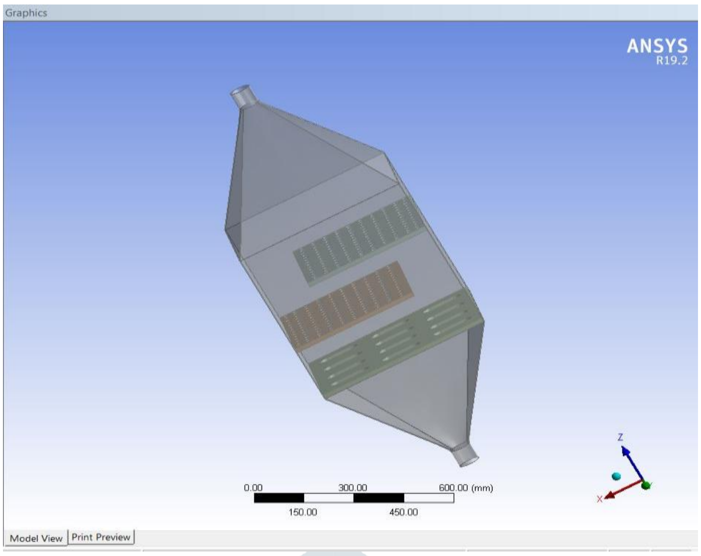

Fig .2. Cabinet dryer 3-D geometry

b) Meshing

In order to analyze fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D. The sizing for the mesh is done manually.

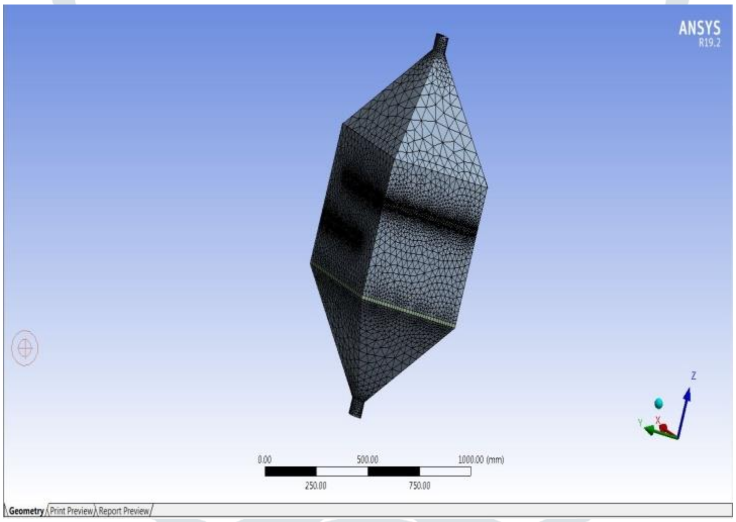

Fig. 3 Discretization of the cabinet dryer geometry

Table 1 Number of elements and nodes

Domain	Nodes	Elements
Cabinet dryer solid base plate	4058	1775
Fluid flow domain	607440	3274612
Insulation solid wall	31408	97477
Porous media tray-1	80772	60650
Porous media tray-2	83400	62905
All domains	807078	3497419

c) Setup and solution

To solve the governing equations, initial and boundary conditions must be defined around the boundary of the system. Specification .since the equations are highly nonlinear, they are not solvable by explicit, closed-form analytical methods. The numerical finite volume method as used in Ansys 19.2 version for the solving the equation on a PC Intel Core i5, 2.7 GHz with 8 GB random access memory. Setting up of flow –simulating computation involves specific boundary conditions, in particular at surface bounding of the domain. In this study various boundary condition was defined as followings:

- Inlet: Air flow rate (2 m/s) was selected Direction of air flow was normal to the air inlet.

- Outlet: assuming gauge pressure-0 at the outlet, Fluent extrapolated the required information from the interior of the drying chamber.
- Porous media: empirical parameters of pressure drop equation and fruit bed porosity and aluminum netting were defined.
- Wall: heat transfer coefficients of the chamber walls and environmental conditions were defined.

VI. RESULT AND DISCUSSION

a) Residual

The residual is one of the most fundamental measures of an iterative solution's convergence, as it directly quantifies the error in the solution of the system of equations. In a CFD analysis, the residual measures the local imbalance of a conserved variable in each control volume. Therefore, every cell in your model will have its own residual value for each of the equations being solved.
In an iterative numerical solution, the residual will never be exactly zero. However, the lower the residual value is, the more numerically accurate the solution. Each CFD code will have its own procedure for normalizing the solution residuals. It is best to check your code's documentation for guidance on appropriate criteria when judging convergence.

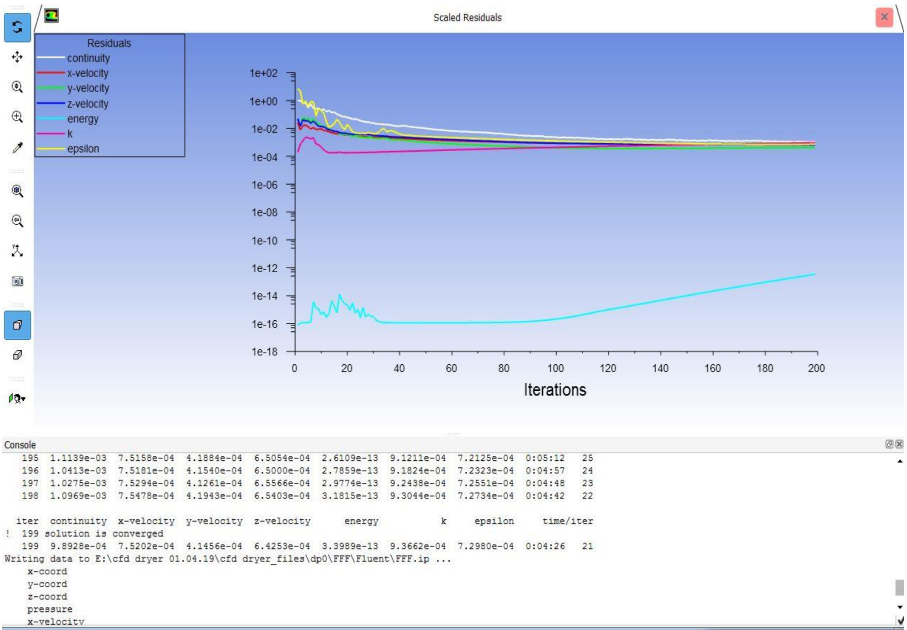

Fig. 4 Residuals of the cabinet dryer in Ansys Solver.

Figure 4 shows that residuals are converged at iteration no 199 which means the last two consecutive grid values are not having a large difference.

b) Temperature contour

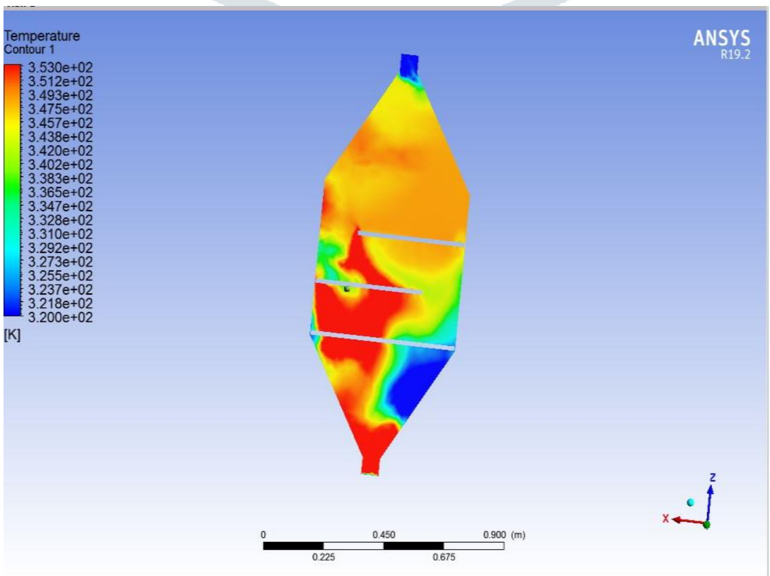

Fig. 5 CFD Post result of the Cabinet dryer Temperature contour.

In the post-processing results as shown above in Fig 5, temperature contour in the first and second trays are not similar. The first tray obtaining the temperature of 353 K While the second tray obtaining approximately 343 K. As it's not a huge difference but it results in the Non – Uniform Temperature Distribution.

c) Velocity contour

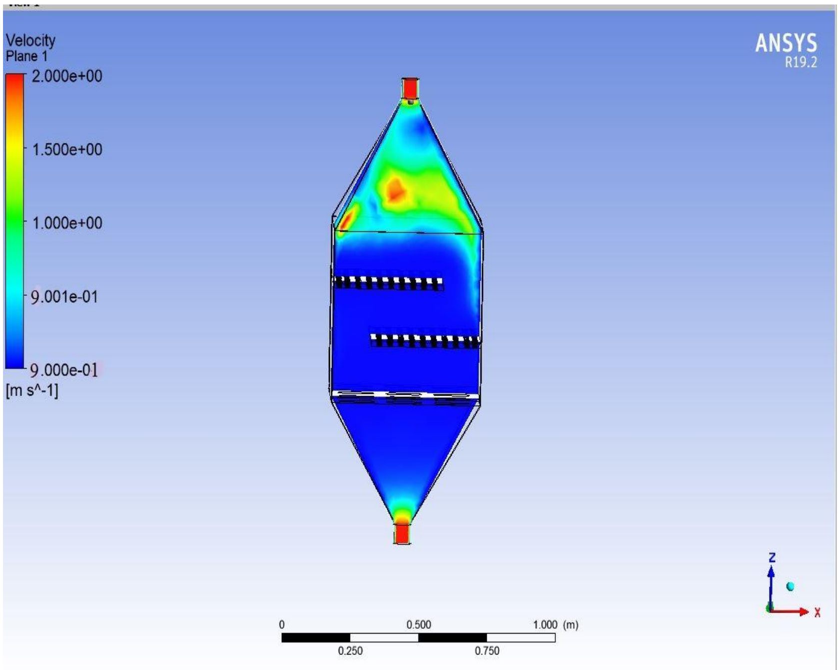

Fig. 6 CFD Post result of the Cabinet dryer Velocity contour.

In the Post-processing results as shown above in Fig 6, Velocity contour, the inlet velocity given is 2 m/s due to divergence in the cross-section of the chamber, inlet velocity drops to approximately 0.9 m/s and maintains this flow rate uniformly across the cross-section of the chamber

VII. CONCLUSIONS

In the present research, CFD analysis for cabinet dryer has been carried out for the velocity and temperature distribution profiles inside the chamber, we obtained a Non-uniform temperature distribution across the trays as well as velocity drops initially at the divergent section and then maintain a uniform flow rate between the trays. Finally, we conclude that

- As the temperature drops between tray 1 (353.15 K) and tray 2 (343.15 K), it is recommended to place the high moisture content food product in the bottom tray and a low moisture content food product in the lower tray for the drying process.
- The velocity drops in the divergent section but getting a uniform velocity (0.9 m/s) at both the trays it is recommended. In the future research process of this dryer we have to redesign the divergent section for the effective velocity.
- This cabinet drying chamber as for as from the CFD results that we obtained it is recommended to use for drying the food products by keeping the different moisture content products at different trays. If we use the same moisture content products in both the trays then the end result will be not efficient, which means the drying of the upper tray will not be more effective while compared to the bottom tray. So, two different kinds of food products of different moisture content one are higher than the other are used for drying inside this cabinet drying chamber.

REFERENCES

[1] Abhay L, Chandramohan VP, Raja VRK (2017) Design, development and performance of indirect type solar dryer for banana drying. Energy Procedia 109:409–416.

[2] Mulet A, Berna A, Borras M, Pinaga F (1987) Effect of air flow rate on carrot drying. Dry Technol 5(2):245–258.

[3] Norton T, Sun DW (2006) Computational fluid dynamics (CFD)—an effective and efficient design and analysis tool for the food industry: a review, Trends Food Sci Technology 17:600–620.

[4] Amanlou Y, Zomorodian A (2010) Applying CFD for designing a new fruit cabinet dryer. J Food Eng 101:8–15.
Adams RL, Thompson JF (1985) Improving drying uniformity in concurrent flow tunnel dehydrators. Trans ASAE 28(3):890–892.

[5] Foster AM, Madge M, Evans JA (2005) the use of CFD to improve the performance of a chilled multi-deck retail display cabinet. Int J Refrig 28:698–705.

[6] Hong SI, Lee H, Hwang I, Seo J, Bitog J, Song (2011) CFD modeling of livestock odor dispersion over complex terrain. 108: 253–264.

[7] Franco ADL, Valeva A, Pena A, Perez M (2011) Aerodynamic analysis CFD simulation of several cellulose evaporative cooling pads used in Mediterranean greenhouses. 76: 218–230

[8] Karathanos VT, Belessiotis VG (1997) Sun and artificial air drying kinetics of some agricultural products. J Food Eng 31(1):35–46.

[9] Incropera FP, DeWitt DP (1990) Fundamentals of heat and mass transfer. Wiley, New York

[10] Yongson O, Badruddin IA, Zainal ZA, Narayana PAA (2007) Airflow analysis in an air conditioning room. Build Environ 42:1531– 1537.

[11] Verboven P, Scheerlinck N, De Baerdemaeker J, Nicolaï BM (2000b) Computational fluid dynamics modeling and validation of the isothermal airflow in a forced convection oven. J Food Eng 43:41–53

[12] Scott G, Richardson P (1997) The application of computational fluid dynamics in the food industry. Trends Food Sci Technol 8:119–124

[13] Mirade PS (2003) Prediction of the air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models. J Food Eng 60:41–48

[14] Xia B, Sun DW (2002) Applications of computational fluid dynamics (CFD) in the food industry: a review. In: Sun DW (ed), CFD Applications in the Agri-Food Industry (special issue) Computers and Electronics in Agriculture, 34(1–3): 5–24

[15] Ver.boven P, Scheerlinck N, De Baerdemaeker J, Nicolaï BM (2000a) Computational fluid dynamics modeling and validation of the temperature distribution in a forced convection oven. J Food Eng 43:61–73.