УДК 631.365.22
A.O. Kravtsova
Glyndwr University Wrexham (Wrexham, North Wales, United Kingdom)

ANALYSIS OF FRUITS DRYING PROCESS USING COMPUTATIONAL FLUID DYNAMICS

У статті викладені результати дослідження процесу переміщення потоку сушильного агента через шар матеріалу (яблука) під час його конвективного сушіння.

Ключові слова: *конвективне сушіння, фрукти, сушильний агент, шар матеріалу, аеродинаміка, моделювання процесу.*

Drying is one of the most commonly used methods of preservation and processing of fruits and berries. In this relation, particular importance of the research is to develop methods of drying, which provide high quality finished product, creating conditions for more complete processing of fruits, reduce losses, as well as automation and mechanization of the drying process.

The essence of the drying process is summing up the heat to the product that is dried, the allocation of water from its inner layers to the surface and moisture removal in the form of steam in the environment. Convective drying method is the most widely used and is used in majority of drying installations worldwide.

Successful drying is a uniform heating of the product to the maximum possible temperature with minimum heat applied to the product. When investigating fruits and berries drying, it is important to know the basic laws that characterize this process, including change in temperature of the drying agent, the initial moisture content of the material and its influence on the speed of the process.

Convective drying method is simple and can be regulated different temperature and material. This method is the most common and is a base of most drying installations worldwide. Therefore this work is related to convective drying method. Convective method is based on the supply of heat air to product and water excretion from object together with coolant.

Convective drying process can be carried out by using various kinds of machinery such as spray drier, belt drier, air cabinet, tunnel drier, foam drier, drum dryer, microwave oven, freeze-drier, and fluidized bed. This work will be concentrated on the study of air cabinet for convective drying process.

Computational Fluid Dynamics (CFD) used in food industry can serve better understanding of physical processes. CFD is applicable to specific areas such as refrigerated transport, clean-room design, static mixers and others. Food industry can significantly benefit from use of CFD methods. This analysis tool enhances the understanding basic physics of fluid dynamics and design process and can provide profit to many food manufacturing processes such as drying, mixing, pasteurization, crystallization sterilization and other areas. In this work CFD is used as a tool for analysis of convective process of drying fruits.

Computational fluid dynamics (CFD) uses applied mathematics and powerful computers to model flow situations. In recent years that computational fluid dynamics has been used in the food processing industry. The drying rate is an essential function of air velocity or air flow in the dryer. Therefore, it is very important to know the velocity and air flow inside the drying chamber because it enables us to understand a zone of efficient air velocities for accurate drying process. But air velocity and airflow are very complicated to calculate during drying process sine these sensors are essential in the different locations and directions of airflow. That is why computational fluid dynamics has been applied as a powerful tool to predict air velocity and also airflow during drying.

Kaya et al. (2008) led their research a two-dimensional study of mass and heat transfer of an observed object using a finite difference method on the basis of the convective boundary conditions at surfaces of object. The changeable convective heat and mass transfer coefficients has been examined under the drying. The temperature fields and external flow were predicted with the CFD package Fluent. The effect of the part ratio on the mass and heat transfer has been studied. The temperature fields and external flow round the product, moisture and heat transfer inside the object were first analyzed. The modification of the convective heat transfer coefficients alongside the side of the moist object was determined. The convective coefficients at the left-side-wall of object have been found to be higher than at other sides due to peculiarities of air upstream. It has been discovered that higher aspect ratios guarantee longer drying period on the contrary to smaller aspect ratios.

The numerical and experimental study on mass and heat transfer during drying of kiwi fruits has been performed by Kaya et al. (2006). The influence of versatile drying terms such as air temperature, velocity and relative humidity on drying process of qualities of kiwi was explored in the experimental part. Experiments were implemented under the following preconditions: relative humidity values at 25%, 40%, 55% and 70%; velocities drying air at 0,3; 0,6; 0,9 m/s; temperatures air at 25, 30, 40, 50, 60°C. Numerical part of the research was aimed at analysis of the external temperature fields and flow using a CFD package. The temperature fields and external flow around the investigated object (25 mm, 25 mm, 45 mm) under the drying process were predicted. The airflow velocity increase showed the growth of the moisture transfer coefficients and local heat. Besides, the moisture and temperature fields inside the object were studied at different periods using the code developed by the researchers. The time-dependent moisture and temperature distributions for various circumstances were also obtained thanks to the utilization of the code devised to inquire heat and mass transfer aspects in the kiwi fruits. To complete the research, experiments have been collated with the numerical findings. Outcomes of two investigation approaches appeared to be in great agreement.

Mohan at al. (2010) studied 3-D numerical model performed for the prediction of transient moisture and temperature distribution in a rectangular object during the process of convective drying. The heat transfer coefficients of the moist object have been calculated using (CFD) code. The mass transfer coefficients were received by analogy from the concentration and thermal boundary layer. The effect of temperature and velocity of the drying hot air on an object has been analyzed. The optimal time of drying process has been predicted for variety of air input temperature, moisture content and velocity. Air flow velocity appeared to have significant influence on drying mechanism. The increase of the air flow velocity led to the increase of drying rate. This research has shown that about 40% of drying process time was saved due to increase of the temperature from 313 to 353 K. A constant rate drying period was not found at different air flow velocities and drying temperatures. The obtained results this work have shown that the proposed model with consideration of variable diffusion coefficient, transfer coefficient and convective boundary condition is suitable for predicting accurate moisture and temperature distribution of a moist object at differing velocity and temperature of drying air.

The development of a theoretical model which describes the simultaneous transfer of momentum taking place in a convective vegetable drier where air flows under turbulent conditions round a vegetable sample has been researched by Curcio et al. (2008). The proposed model is able to depict the real behavior of convective dryers over a wide range of fluid-dynamic and process terms. The system of non-linear differential equations which modeled the behavior of vegetable sample in a drier has been solved by using finite elements method. It was proved that the air characteristics had influence mostly when external resistance to mass transfer is the rate controlling step. It was observed that the model also imitated the moisture profiles at different periods. Simulations results have shown good agreement with experimental results in terms of time evolutions of food temperature and moisture content.

Erriguible et al. (2006) published a paper on the analysis of the boundary conditions during convective drying. The problem solving method of this research was a conjugate transfer between a surrounding and porous medium. A piece of porous medium under convective drying using a coupling method between computational fluid dynamics software and a porous medium code was simulated.

The transfer coefficients were used for description of transfer between a porous medium and its surroundings. The parsing of the moisture content profiles and temperature and studying of the interfacial transfer coupled have demonstrated the influence of the edges. The constant rate period heat is not quite applicable for evaporation with a portion for increase of temperature inside the product. The physics involved during the convective drying of capillary porous medium was described.

In 2010 Amanlou and Zomorodian (2010) published a paper about a new dryer chamber which was designed and constructed using computational fluid dynamics software. Different geometries and volumes of drying cabinet of convective dryer were predicted experimentally and theoretically using CFD. The influence of differing constructions of drying chamber on dryer performance was investigated. Three air temperatures about 20, 30 and 40°C and velocities 1, 2 and 3 m/s were analyzed. In addition, heat transfer coefficients of the chamber walls, specific heat and density, thermal conductivity and environmental conditions were taken into consideration. The distribution of air temperature and flow of hot air taking into consideration the general operating conditions was obtained. The general k-ε model with empirical data from CFD (porosity and trays pressure drop) demonstrated exact results. The most suitable sketch with acceptable temperature distribution and uniform flow was used for conducting the experiments. The results of experimental established that the new chamber of convective dryer showed an uniform allocation of air temperature and velocity along chamber. Comparison of CFD analysis data with experimental data has shown a good correlation coefficient of 86,5% air velocity and 99,9% for air temperature. In this work computational fluid dynamics was a powerful tool for systemic study and analysis of drying chamber.

The study on drying characteristics of apple slices during drying was conducted by Meisami-asl at al. (2009). Three thicknesses of slices under five different air temperatures and drying behaviors of apple slices was investigated. Thin kinetics drying of slices was experimentally analyzed in a convective air dryer and the mathematical modeling of drying was realized with using layer convective drying models. Drying performance of apple were fixed using air velocities at 0,5 and thickness of layer slices 2, 4, 6 mm and heated outward air at temperatures 40, 50, 60, 70 and 80°C. In this paper influence of slice thickness on the convective drying characteristics, quality of dried apples and time of drying were studied. The main results this work showed those shorter drying times were caused by increase of drying air temperature. Kuriakose et al. (2010) performed their research on the simulation of the spray dryers for food materials with using CFD. This work has shown an important application of CFD of drying to envisage the complex flow area. The effects of turbulence in the drying chamber have been reviewed. Modeling velocity of particle, temperature, the particle position and residence time during spray drying were analyzed.

Mirade and Daudin (2000) researched a paper concerning possibility ability to use of computational fluid dynamics (CFD) to for prediction of airflow patterns in a dryer by comparing experimental with numerical results velocity data. The useful information about circulation of air within modern sausage dryer by using predicted of velocity fields was provided. The 3D calculations by using general-purpose CFD were performed. FiA five 3D meshes appropriating were related to the five experiments built in the dryer. The several boundary conditions were composed at the outlet and inlet of the fluid area studied. The turbulent kinetic energy, velocity magnitudes, dissipation rate of the kinetic energy in the inlet area was determined. The air velocity was calculated by using the blower airflow rate in the apparatus. The principal turbulence flow was taken into using the application of the k-ε model where the standard function of wall was applied. The calculations results were compared at a few thousand of points, the measure velocities with calculations results velocities. Quantitative and qualitative comparisons demonstrated that measurements values agreed were in agreement with simulated results although certain dissensions for specific terms of airflow have appeared. The measurement errors of measurement were identified by comparisons revealed. Besides, qualitative comparison displayed that calculations are ability to could reflect the influence of filling level on flow patterns. The error of measurement error in areas where the flow was being in a horizontal direction was detected by quantitative comparison. Therefore, the air velocity and airflow pattern in the drier has been predicted using computational fluid dynamics modeling.

The air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models was prediction and analyzed by Mirade (2003). Two-dimensional CFD model with unsteady model (i.e. a time-dependent boundary conditions) was using in this research. The homogeneity of the allocation of the airflow velocity in modern meat dryer is showed for a few high and low levels of the air ventilation cycle, and sinusoidal and linear forms of cycle. The alterations structure of air flow appropriate to dissymmetry taking place in the cycle of ventilation was estimated.

The present work demonstrated that computational fluid dynamics approach could be useful for estimating the functioning of modern dryers for meat. The results confirmed the industrial comments concerning functional efficiency of air ventilation system and the necessary for controlled regulation of the dryer ventilation cycle. This paper highlights the benefit of CFD for effectiveness of the functioning of food industry.

Mathioulakis et al. (1998) performed a research on industrial tray air dryer which was created for the convection drying of several types of fruits. The air movement in the convection chamber was modeled using computational fluid dynamics. The air velocities and pressure profiles inside the convective drying chamber around the product were established by computational fluid dynamics and also a shortage of spatial homogeneity of the air velocities around the fruits was detected. Drying tests of some fruits were implemented and the result of drying tests showed the determined fraction of weight loss. The non-uniformity and differences in aridity degree in some trays with fruits was detected in explicit areas of the drying chamber. The dependency of consequents from tray to tray and the non-conformity in convection drying in several zones was analyzed using CFD.

The air circulation is inconstant and was explored to be more intense in the certain areas. Comparison information achieved from the convection drying tests and of information obtained using the CFD demonstrated a ponder able correlation between the velocity of air and drying rate.

Therefore, there is a need for studies of various aspects of drying to improve drying conditions and technological parameters. The outcomes from previous researches have shown that computational fluid dynamics (CFD) can be applicable and useful tool for analysis and predicting the air flow behavior with various velocities, temperature and residence time. Moreover, CFD can be used for study behavior of fruits materials as well.

The turbulence model is used for describing an influence of turbulence on flows. The two-equation turbulence models present a compromise second-order closure models, which need computing resources since they account the algebraic models that assume local equilibrium and the non-isotropic nature of the Reynolds Stresses. The *k-ε* model is the most widely used one out of the existing two-equation models in engineering problems in combination with functions which bridge the buffer and viscous layers. In the present work, the standard *k-ε* turbulence for predict the air flow field in the dryer studied (Jones and Launder, 1972).

This type of turbulence model assumes conservation of kinetic energy, *k*, and its appropriate dissipation rate, *ε*, can be expressed:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \qquad (1)$$

and

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon, \qquad (2)$$

where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients; G_b is the generation of turbulence kinetic energy due to buoyancy; Y_M is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; C_1, C_2, and C_3 are constants; σ_k and σ_ε are turbulent Prandtl numbers for k and ε. Accordingly; S_k and S_ϵ are user and need source terms.

The transfer of heat and mass in the gaseous phase are presented by the equation:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + p)] = \frac{\partial}{\partial x_j}\left(k_{eff}\frac{\partial T}{\partial x_j} + u_i(\tau_{i,j})_{eff}\right) + S_h, \qquad (3)$$

where *E* is the total energy; k_{eff} is the effective thermal conductivity; $(\tau_{i,j})_{eff}$ is the deviatoric stress tensor, defined as

$$(\tau_{i,j})_{eff} = \mu_{eff}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\mu_{eff}\frac{\partial u_k}{\partial x_k}\delta_{ij}. \qquad (4)$$

For the standard *k-ε* model, the effective thermal conductivity is

$$k_{eff} = k + \frac{c_p \mu_t}{Pr_t}, \qquad (5)$$

Where Prandtl number is 0.85, *k* is the thermal conductivity.

The convective dryer investigated in this study was tray air dryer. This dryer was studied for operation with fruits. Heated air was supplied through entrances into the middle chamber and removed by one outlet. The product was placed into shelf along all dryer.

In this study the distribution of air flow during convective drying process was important. The air flowed through a small opening placed along of two walls across chamber. Then the flow passed through the trays with fruits in the Y direction before quitting the drying compartment through one of the outlets which were placed in to the top centre of one end the dryer, and passaged in the Y direction. This investigation has been concentrated on the accurate representation of the two-dimension air flow dynamics. Due to the fact that the basic flow develops in two directions within chamber, the use of a two-dimensional grid is indispensable. As a outcome, the dependency variables of the task are the two components of velocity such as U and V in the X and Y directions.

In this investigation Fluent was used for study convective drying process. Fluent is a computer program for modeling fluid flow and heat transfer. FLUENT uses unstructured mesh technology. Adaptation of the computational mesh allows get an exact solution for different oblasts. FLUENT is one of the best packages in multiphase modeling. Various features of the program can get the deepest product information. Quality of computer simulation results depends on the quality of construction of the calculation mesh. Preprocessor GAMBIT allows you to quickly create and edit geometry of the investigated processes. GAMBIT has a single interface for creating geometric models and meshing. Different CFD problems require the construction of different types of mesh. GAMBIT has a powerful mesh generator that allows to creation of different mesh types.

That's why GAMBIT was used to construct a 2D model. GAMBIT was used for creation of the drying chamber geometry, meshing the geometry and assigning appropriate zone to the boundaries.

Boundary conditions debugging a flow-simulating computation contain special boundary conditions, especially at surfaces confining the domain. In a given case the following boundary conditions were put to use:

1) At the inlet, velocity inlet boundary conditions are used to define the velocity of flow, along with all appropriate flow properties, at flow inlets. The total flow properties are not fixed, so they will rise to different values. This is indispensable to provide the prescribed distribution of velocity. A velocities-inflow condition was assumed with velocities 0,5 m/s and two air temperatures (50°C and 70°C) were selected.

2) At the outlet, a boundary condition was imposed to allow outlet pressure stay as the domain solution. Pressure outlet conditions demand the specification of a static pressure at the outlet boundary. All other flow quantative characteristics are extrapolated from the interior.

3) At surfaces bounding the domain of chamber, including those bounding trays, a wall shear stress condition was assumed.

Simulation conducted to detect the real air distribution in drying chamber, determine the influence of velocity and temperature of drying air on the process of drying fruits and on the kinetic of drying. The main factors affecting the drying process such as the temperature of drying agent and the air flow rate were analyzed. Under various drying air conditions, moisture ratio and drying depending on drying time was compared. The uniformity of convective drying also evaluated according to location a piece of apple on the shelves considering the heat transfer coefficient.

Fig. 1 is illustrated the changes of heat transfer coefficient depending from ordinal numbers of shelves and placing fruit on the shelf. On the figures traced the gradual decrease of the heat transfer coefficient to the center and then gradually increase to the next side. That maximum coefficient of heat transfer is observed in the fruit placed on both edges of the shelves, and the minimum is in slices which are in the middle of the shelves.

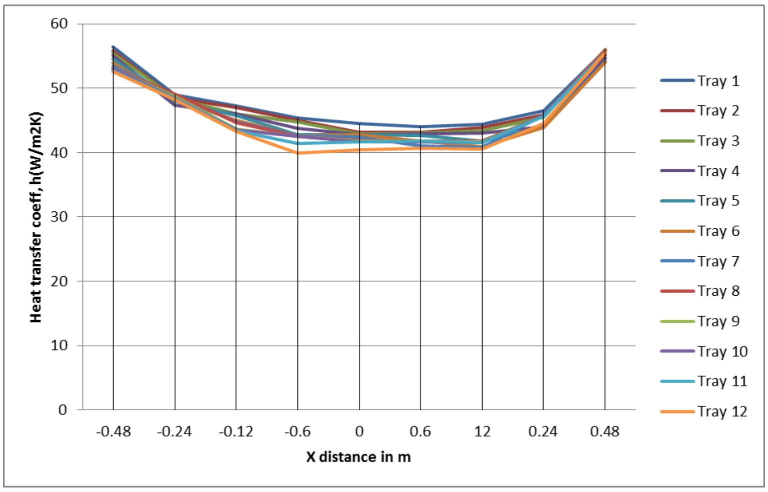

Fig. The heat transfer coefficient is depending from ordinal numbers of shelves and placing fruit on the shelf at temperature 50°C

Conclusions

The quality of dried fruit products are improved with the reduction of drying time, reduction duration of thermal action on the product and depends on the uniformity of heating the material to minimize moisture during drying. Therefore, minimum thermal effects on fruit will have reduced energy consumption in the drying process, high-quality appearance and nutritional value of the product.

Uniform heating of wet fruit during a short time is one of the main aims of this work. Modeling movement of hot air and temperature distribution in the material when heated was modeled using Ansys CFD package. The temperature distribution in the material and predicted drying time modeling was controlled using CFD simulation.

The convective air dryer for agricultural product was investigated in this study. This dryer was studied for operating for apple slices with thickness 6 mm. Two modes of drying using different temperatures of 50 and 70°C and the speeds 0,5 m/s were simulated. The simulations of two drying modes were performed in air dryer to investigate the effect of air velocity, temperature and drying time on the drying kinetics of fruits.

This simulation has shown the differences between genuine air distribution and theoretical scheme of air circulation in the dryer. As it was detected, this fact has considerable consequences and important influence for the homogeneity drying fruits and drying rate.

A 2-D model of heat and mass transfer was developed to study the behavior of temperature and moisture content of a moist object and inside drying chamber. The variables of heat transfer coefficient in the 2-D model are calculated by CFD simulations. The following basic conclusions are drawn:

– It is observed that with the air temperature increase, the drying rate of apples is increased. Convective drying time is decreased about 25% with temperature increase of 20°C at air velocity 0,5 m/s.

– The CFD simulation results showed non-uniformly distribution of air velocity and temperature inside chamber and that there is one area of low air velocity.

– The main result of this paper is to set up a simulation model of convective drying. Simulation by Computational Fluid Dynamics proved to be a effective optimization tool in order to avoid costly and unnecessary experiments, to predict the drying time and to improve the design of dryer. Similarly created model can be used for other types of fruits and vegetables. The drying chamber for many food products will be optimized during minimum time using this way of model. This can reduce the cost of experimentation and shorten the time. From the obtained results, it can be concluded that the various models of convective dryers can be examined using this type of modeling that can lead to the identification of the real problems that exist and help solve them.

– 2D CFD simulation does not reflect the full picture of the distribution of air in the chamber although the results in most of the camera. That why it is interesting to perform the further study of air distribution in chamber and provide more information of the flow field in 3D. Also, this method of simulation does not show changes of volume size and deformation of fruits during drying, although showing the evaporation of water from fruits. This problem can be solved using UDF code. Another important point for further work can be cost of running a dryer. A cost of running a dryer depends on amount of heat that is required for heating drying zone, analysis of costs in terms of the requirement of energy for different methods of drying.

1. Amanlou Y., Zomorodian A. (2010) Applying CFD for designing a new fruit cabinet dryer. Journal of Food Engineering, Vol. 101, pp. 8–15.
2. Erriguible A., Bernada P., Couture F. and Roques M. (2006) Simulation of convective drying of a porous medium with boundary conditions provided by CFD. Chemical Engineering Research and Design, Vol. 84, pp. 113-23.
3. Jones W.P., and Launder B.E. (1972) The Prediction of Laminarization with a Two-Equation Model of Turbulence. International Journal of Heat and Mass Transfer, Vol. 15, pp. 301-314.
4. Kaya A., Aydın O. and Dincer I. (2008) Experimental and numerical investigation of heat and mass transfer during drying of Hayward kiwi fruits, Journal of Food Engineering, No.88, pp. 323-330.
5. Kaya A., Aydın O. and Dincer I. (2006) Numerical modeling of heat and mass transfer during forced convection drying of rectangular moist objects, International Journal of Heat and Mass Transfer, No.49, pp. 3094-3103.
6. Mathioulakis E., Karathanos V.T., Belessiotis V.G. (1998) Simulation of Air Movement in a Dryer by Computational Fluid Dynamics: Application for the Drying of Fruits. Journal of Food Engineering, Vol. 36, pp. 183-200.
7. Meisami-asl E., Rafiee S., Keyhani A., Tabatabaeefar, A. (2009) Mathematical Modeling of Moisture Content of Apple Slices (Var. Golab) During Drying. Pakistan Journal of Nutrition, Vol. 8, Issuer 6, pp. 8–15.
8. Mirade P.S., Daudin J.D. (2000) A numerical study of the airflow patterns in a sausage dryer. Drying Technology, Vol. 18, pp. 81-97.
9. Mirade P.S. (2003) Prediction of the air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models. Journal of Food Engineering, Vol. 60, pp. 41–48.
10. Mohan V.P., Talukdar P. (2010) Three dimensional numerical modeling of simultaneous heat and moisture transfer in a moist object subjected to convective drying, International Journal of Heat and Mass Transfer, No.53, pp. 4638-4650.