[1.]Izudin DELIĆ, [2.]Amel MEŠIĆ

CFD ANALYSIS OF FLUID STREAMING IN ROTARY DRYER

[1-2.] Faculty of Mechanical Engineering, Tuzla, BOSNIA & HERZEGOVINA

Abstract: The dryer in which drying of material is being done by rotation moving of the drying stands around which hot air is streaming represents an innovative technical solution. Counter-streaming movement of the rotor with drying stands through the stream of hot air enables a higher drying intensity. In order to get a clearer picture of the hot air streaming among the drying stands and the effects of the individual components on the streaming of fluid through rotary dryer, it is necessary to perform a detailed CFD analysis.
Keywords: Rotary dryer, streaming, drying, CFD

INTRODUCTION

Drying is the simplest and most natural way of all procedures for food conservation, by which food freshness is preserved given that most of free water is extracted from it. Drying may be carried out naturally on the sun (direct drying on the sun) or ventilation and heat may be added in order to accelerate drying process (electro, gas or diesel drying plants). Dry, warm air is ideal for drying but only if air temperature is strictly controlled. Convective dryer for fruits and vegetables is based on innovative solutions and complete control of all drying parameters (temperature, humidity and air circulation rate). During the first half of 2015 a prototype of innovative dryer for fruits and vegetables was successfully designed. In the dryer of Eco –Rotary type biomaterial is dried at low temperatures (max. T=55-70°C) and low humidiy (RH= 20-25%).

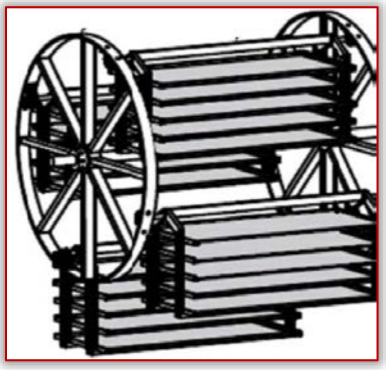

Figure 1. Representation of the rotary dryer interior

The operation principle of the innovative solution “Eco-Rotary Dryer” is reflected in the radial movement of the stands with the material for drying. On the other hand, pre-heated air is blown into the chamber to circulate around the material that is being dried in the opposite direction of the movement of the rotor with the drying stands. This counter-streaming movement enables the increase in the drying intensity as well as moving material for drying through various thermal zones in the drying chamber.

In order to determine the nature of the air streaming around the rotor and the drying stands with material for drying, a detailed CFD analysis was performed.

NUMERICAL MODEL

Defining boundary and starting conditions represents an essential step in conducting of the numerical simulation. Therefore, while defining boundary conditions, the specific features of the process were taken into account. For that purpose, four regional boundaries were defined:

- Fluid entrance (the boundary where entrance velocity and temperature are set up);
- Fluid exit (the boundary where it is defined for all fluid to exit the domains);
- Wall (the boundary where the value of thermal flux is defined);
- Internal construction (the boundary which is defined as being adiabatic);

On the basis of the data obtained from the thermal calculation, the following starting conditions have been defined:

Tabel 1. Starting data for calculating air streaming in the rotary dryer

Size	Value	Dimension
Air velocity	5	m/s
Air temperature	70	°C
Enviromental temperature	20	°C
Thermal flux (floor)	17.65	W/m^2
Thermal flux (ceiling)	9.39	W/m^2
Thermal flux (side walls)	17.07	W/m^2
Thermal flux (back wall)	17.21	W/m^2
Thermal flux (doors)	53.40	W/m^2

Spatial discretization was done by using polyhedral net while for representation of the results of the numerical simulation we have chosen several plains that are suitable for representing streaming of the fluids. In particular areas, the net optimatisation has been done with the aim to reduce the number of cells depending on the needs and wanted accuracy of the simulation results, which can be seen in the picture.

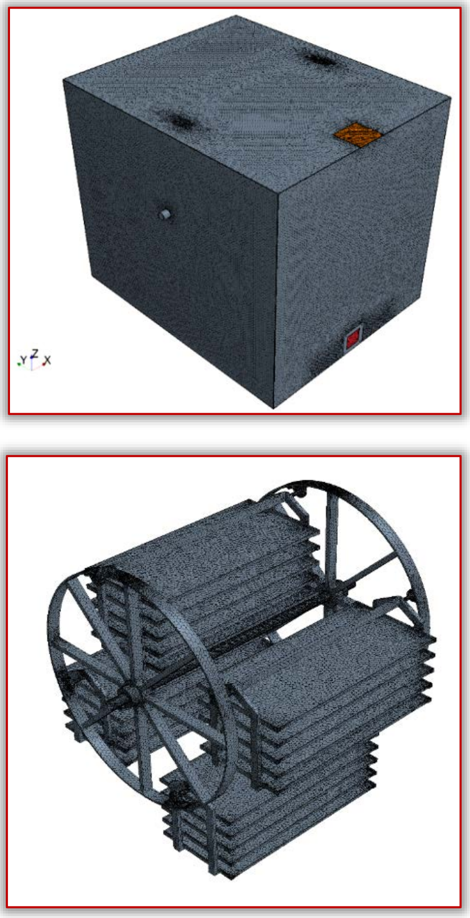

Figure 3. Numerical network of the model with defined boundaries

CFD ANALYSIS OF THE AIR STREAMING IN THE CHAMBER OF THE ROTARY DRYER

The Figure below shows the streaming of the fluid in the drying chamber where the nature of the air streaming around the stands with material for drying could be identified. It could be noticed that there is an intensive turbulent and almost symmetric streaming in the lower part of the chamber while the streaming in the upper part of the chamber is more even.

On the basis of the Figure below, it can also be noticed that the drying stands are not evenly streamed by the hot fluid. Moreover, the lower stands receive the most streamed fluid while those stands located just above the entrance of the air into the chamber get the least amount of the streaming hot fluid.

The streaming of the fluid could be noticed in more details in the characteristics fields that we have chosen for representation of the results.

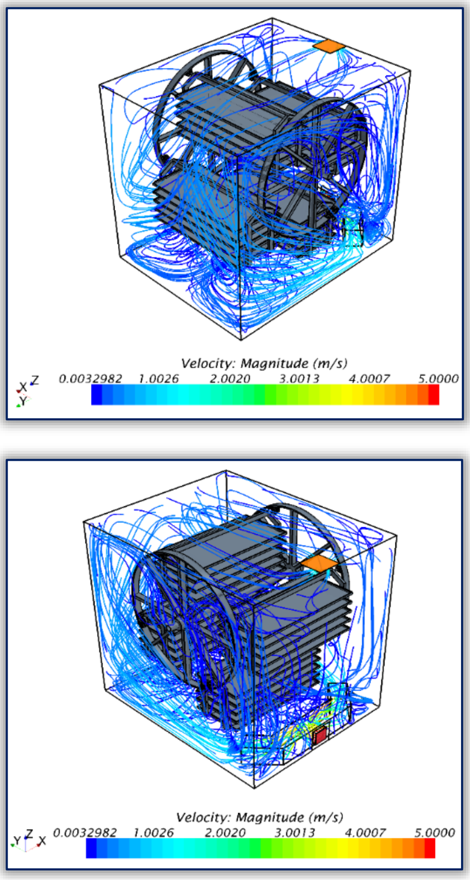

Figure 4. Representation of the fluid streaming in the drying chamber

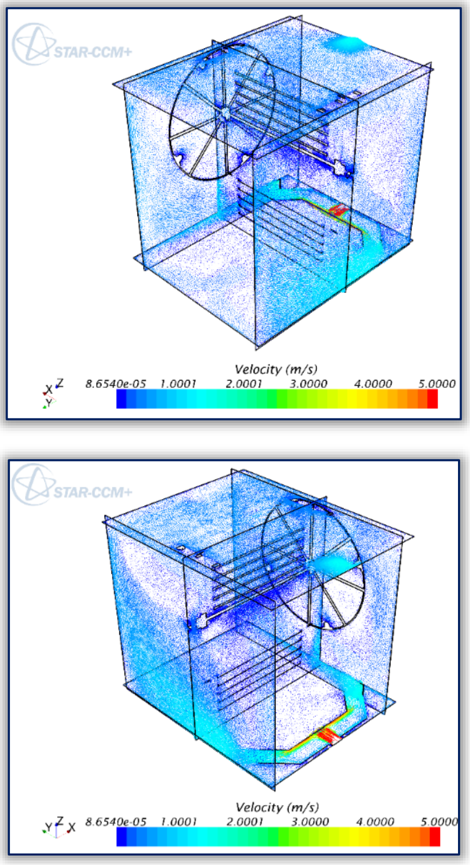

Figure 5. Representation of the vector-velocity fields in several cross-section lines

It is clearly visible that there is a significant decrease in the streaming of the fluid through distribution channels used to supply the hot air. Observing the streaming circuit in the Figure 5, there are visible zones of the intensive whirling of the air current alongside the chamber wall, as well as weak air streaming in the fields in the upper parts of the internal construction. Having in mind the distribution of the stands, it is evident that the stands will not be supplied with hot air with the even speed and, therefore, the drying process will be of different intensity in some zones.

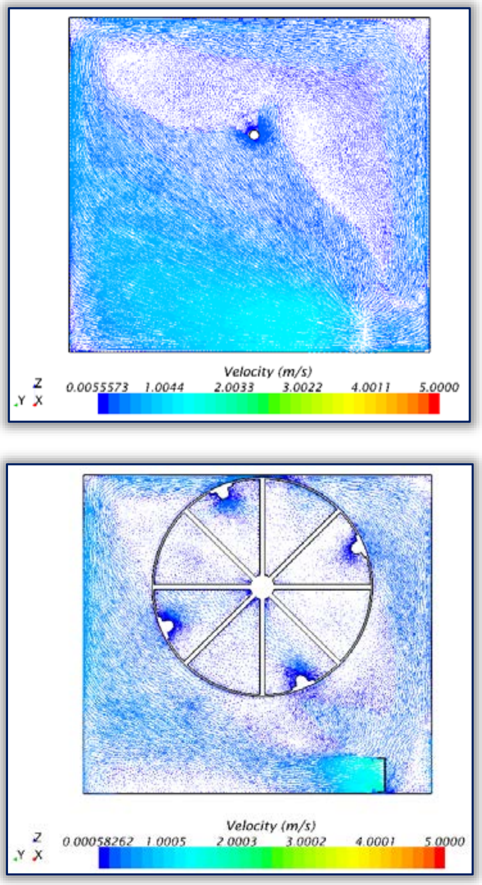

Figure 6. Left: Vector field of the speed alongside side wall, Right: Wheel.

It can also be identified that there are some places where there is need to position suitable fluid routers in order to balance even streaming around all the drying stands and thus improve the drying process.

Besides the influence of the speed of the air streaming on the drying process, there is also a particular influence of the temperature. Therefore, the temperature circuit in the chamber has also been shown. The analogy between thermal and speed field, where the highest thermal values are seen in those zones in which the speeds are the highest and the lowest thermal values are visible in the zones of the least speed values. In the thermal field, there is also visible thermal flux towards external surfaces. Uneven thermal distribution is also a consequence of the size of openness of the regulation flap that represents an additional distribution to the thermal uniformity. The orientation of the distribution channels is such that enables the hot air to hit the side walls with maximal speed while there is visible loss of the streaming energy.

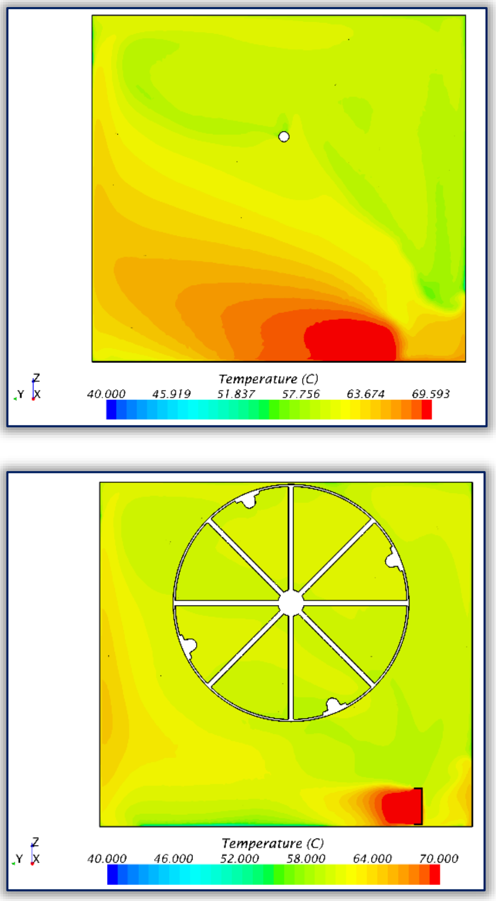

Figure 7. Thermal distribution in the chamber

CONCLUSION

Based on the results of the simulation, it can be concluded that the air streaming inside the chamber of the rotary dryer does not have a suitable feature for a biggest interval of the process parameters. In addition, the analysis of the air streaming has shown that the stands for drying do not receive even amount of the hot streaming fluid. There are visible places where directors of the air should be installed in order to decrease the streaming losses and to evenly direct the flow of the fluid towards the drying stands. A regulation blade should be mounted at the feeding channel of the hot air in order to direct the hot air depending on the required conditions of the drying process.

These changes would increase the drying intensity, lower duration of the drying process and increase the product competitiveness. By controlling drying parameters (temperature, humidity and air circulation rate) technological conditions in compliance with the required parameters of drying kinetics of many kinds of fruits, vegetables, forest fruits and medical herbs may be provided.

Note

This paper is based on the paper presented at 13th International Conference on Accomplishments in Mechanical and Industrial Engineering – DEMI 2017, organized by University of Banja Luka, Faculty of Mechanical Engineering, in Banja Luka, BOSNIA & HERZEGOVINA, 26 - 27 May 2017.

References

[1] Misha S., Mat S., Rosli M.A.M., Ruslan M.H., Sopian K., Salleh E, (2013). The Prediction of Drying Uniformity in Tray Dryer System using CFD Simulation. International Journal of Machine Learning and Computing, Vol. 3, No. 5, p. 419-423.

[2] Akinola A. A., Abubakar M., Kehinde A. (2012). Analysis of a Solar Dryer Box with Ray Tracing CFD Technique. International Journal of Scientific & Engineering Research, Volume 3, Issue 10, ISSN 2229-5518.

[3] Misha S., Mat S., Rosli M.A.M., Ruslan M.H., Sopian K., Salleh E. (2015). Simulation of Air Flow Distribution in a Tray Dryer by CFD. Recent Advances in Renewable Energy Sources, Proceedings of the 10th International Conference on Energy & Environment (EE '15), p.126-135.

[4] Mirade P.S., (2013). Prediction of the air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models. Journal of Food Engineering, vol. 60, no. 1, pp. 41-48.

[5] Norton T., Sun D.W., (2016). Computational fluid dynamics (CFD) – an effective and efficient design and analysis tool for the food industry: A review. Trends in Food Science & Technology, vol. 17, no. 11, pp. 600 -620.

[6] Amanlou Y., Zomorodian A., (2010). Applying CFD for designing a new fruit cabinet dryer, Journal of Food Engineering, Volume 101, Issue 1, Pages 8-15.

[7] Predstavljena prva eko rotaciona susnica u svijetu, http://kameleon.ba, accessed on 2017-02-02.

ISSN: 2067-3809