Fluid flow simulation of industrial fixed bed mixed-flow grain dryer using *k-ω* SST turbulence model

Anderson Rodrigo Visconcini*, Cid Marcos Gonçalves Andrade, Alexandre Marconi de Souza Costa

(*Post-graduation Department of Mechanical Engineering, University of Maringá, Av. Colombo, 3790-Bloco 14-Sala 24 87020-900, Maringá-PR, Brazil*)

Abstract: The fluid dynamics analysis on industrial, mixed-flow grain dryers with fixed bed using computational techniques is necessary to assist the design of such equipment contributing to cost reduction in the dryer projects and agricultural drying operations that involve the production of grains in the world. This study presents a Computational Fluid Dynamic (CFD) solution for air flow analysis in an industrial dryer. The air flow at the inlet and outlet of the dryer was investigated using the *k-ω* SST turbulence model. The dryer region with soybean was considered as a laminar porous medium flow in the permanent and isothermal regime, having for the model a simplified geometry with the tower considered as a porous medium and the air inlet and outlet as a turbulent fluid domain. The flow was treated with a permanent and isothermal regime. Dryer flow and pressures were used according to design parameters. To validate the *k-ω* SST turbulence model, the velocity profile at the dryer inlet was obtained experimentally, which presented results with good agreement between the numerical and the experimental model. The model obtained satisfactory results of the computational validation of the air flow in the dryer with good convergence requiring a minimum of computational effort, being suitable for the simulation of industrial-scale dryers, as to its air flow through a tower, operating with fixed bed soybean in the steady and isothermal regime.
Keywords: computational methods, fixed bed, porous medium, soybean dryer, turbulent flow
DOI: 10.25165/j.ijabe.20211402.5321

Citation: Visconcini A R, Andrade C M G, Costa A M S. Fluid flow simulation of industrial fixed bed mixed-flow grain dryer using *k-ω* SST turbulence model. Int J Agric & Biol Eng, 2021; 14(2): 226–230.

1 Introduction

The grain production is a driver of the world economy, generating direct and indirect jobs, as well as supplying a large part of the demand for food needed to support and develop the world's population. To commercialize and industrialize the produced grains, the drying processes play an important role in the control of the production costs, besides enabling better grain storage and transportation quality, since a large part of the moisture is removed in this drying process, which reduces possible grain damage by various physical and biological agents.

To preserve large quantities of grain for long-term storage, mixed-flow grain dryers are increasingly used worldwide[1] and can operate under continuous grain flow and airflow as well as operate in fixed bed batch mode and continuous air flow. Dryer components that are improperly constructed or arranged can cause long-lasting grain and air flow problems in the dryer, thus having inhomogeneous drying conditions with high energy consumption and quality loss due to uneven drying of the grain[2]. Therefore, to avoid uneven drying, it is important to understand the physical phenomena that occur in the dryer, such as air flow through the grain bed that is affected by the geometry of the dryer and the grain drying tower. The goal then is to improve the dryer design and drying process to ensure product quality and minimize energy waste.

The process of developing new technologies for dryers usually goes through the stages of theoretical conceptualization and development of laboratory-scale models, later on, a pilot-scale to validate the proposed new technologies. This development is of utmost importance, however, it is costly in terms of time and engineering costs, limiting its development or burdening the dryer's construction costs. Computational simulation capabilities can be used as an alternative to contribute to the development of grain dryer technology, for example, it is through Computational Fluid Dynamics (CFD) that one can assist with pre-construction dryer steps, reducing the need for many pilot-scale constructions, which also reduces development time and cost.

Much of the published research on mixed flow drying has focused on increasing dryer performance and preserving product quality through improved dryer control[3,4]. In the steady-state approach, we can cite the work of Kowalski and Pawlowski[5] that presents a mathematical model for stationary and intermittent convective drying. A mathematical model for mixed flow grain dryers was developed by Khatchatourian et al.[6] to simulate dryer performance with and without air recirculation. We can also cite Cao, et al.[7] who developed a two-dimensional simulation model and investigated the influence of dryer body geometry and drying tower height on equipment performance. However, for the application of more efficient CFD techniques in mixed-flow dryers, a compartmentalized analysis is necessary, subdividing the phenomena such as particle flow, air flow, heat transfer and mass, investigating them separately. Finally, sub-models must be combined for a comprehensive drying process model[2].

Initial studies on air flow in grain beds were carried out in the 1950s by Matthies[8] who studied the pressure drop of stored grain

Received date: 2019-08-08 **Accepted date:** 2020-03-12
Biographies: Cid Marcos Gonçalves Andrade, PhD, Professor, research interest: applied Science, engineering and technology, Email: cidmga@yahoo.com.br; **Alexandre Marconi de Souza Costa,** PhD, Professor, research interest: applied Science, engineering and technology, Email: amscosta@uem.br.
***Corresponding author: Anderson Rodrigo Visconcini,** MSc in Thermal Science, research interest: applied Science, engineering and technology. Department of Mechanical Engineering, University of Maringá, Maringá-PR, Brazil. Tel: +55-44-30115849, Email: avisconcini@hotmail.com.

during ventilation and later in the 1960s the first simulations of the grain flow behavior in beds storage with air ventilation emerged did by Brooker[9] accompanied by pressure drop measurements. Already in the 1970s and 1980s, simulation models for airflow and pressure drop in grain beds were published by Thorpe and Hunter[10], Smith[11] and Hunter[12]. After many years, airflow distribution in mixed-flow dryers has come into the focus of research and development. Olesen[13], Cenkowski et al.[14], and Sun and Arnauld[15] experimentally investigated and simulated grain bed air flow distribution between inlet and outlet zones in mixed-flow dryers. To solve the system of differential equations, Cenkowski et al.[14] used the finite element method and compared the calculated results with the experimentally determined velocities. The highest air velocities were observed near the lower edges of the dryer air inlet zone, and the lowest velocities in the upper dryer section through which the air passed. Due to recent advances in hardware and mathematical techniques, CFD analysis has been increasingly used to simulate airflow patterns in mixed-flow dryers[16,17]. For three-dimensional (3D) simulations we have the recent works by Rupesh and Jyeshtharaj[18], Prukwarun et al.[19] and Scaar et al.[2] who analyzed the flow behavior of the fixed grain bed treated as a porous medium taking into account the turbulent airflow models in the dryer, resulting in satisfactory CFD simulations to understand the fluid flow dynamics in mixed-flow driers.

The main goal of this work was to develop a CFD solution for simulation and design of drying equipment for fluid flow behavior inside and outside the drying tower in mixed-flow industrial dryers, where the tower is a fixed soybean bed. The porous media model was used for CFD simulation, validating the model with experimental data obtained from industrial equipment operating in a fixed bed regime. Analysis of the behavior of air flow velocity in the dryer fluid zone, as well as in the fixed grain bed, was performed to verify their behavior. The behavior of the pressure loss was also analyzed.

2 Materials and methods

The purpose of this work was to evaluate the fluid flow dynamics of air in terms of its velocity distribution inside a mixed-flow industrial dryer, where the tower is a fixed soybean bed. We can consider the grain tower as a porous medium in which air flows through the tower from the highest to the lowest pressure in a fluid field defined by the inlet and outlet of the dryer.

This dryer operates in a continuous exhaust flow regime by means of 4 exhaust fans at the top of the cooling zone with a flow rate defined by the manufacturer of 180 000 m^3/h with the diameter of 1.26 m exhaust section of each exhaust fan in the dryer top giving an average speed of 10 m/s per exhaust fan. The work pressure in the dryer cooling zone is defined by the manufacturer as −294 Pa. Air enters the dryer via a rectangular section at the base of the heating zone with an area of 7.5624 m^2 which is connected to the dryer furnace that provides the heated air entering the dryer. The work pressure in the dryer heated zone is also defined by the manufacturer as −176.4 Pa. Still, in the heating zone (dryer inlet) there is a subzone that can be used the time for heating, time as tower cooling zone which has a pressure defined by the manufacturer of −215.6 Pa. The air is vented through the dryer's dusting zone, which is composed of a zone outside the drying domain-containing cyclones and geometry that favors the decantation of solid particles from the stream before it is released into the atmosphere. As the dedusting zone and the furnace are not part of the drying domain, they will be disregarded from the study as they do not influence the study domain at its borders. Figure 1 illustrates the arrangement described above for the dryer.

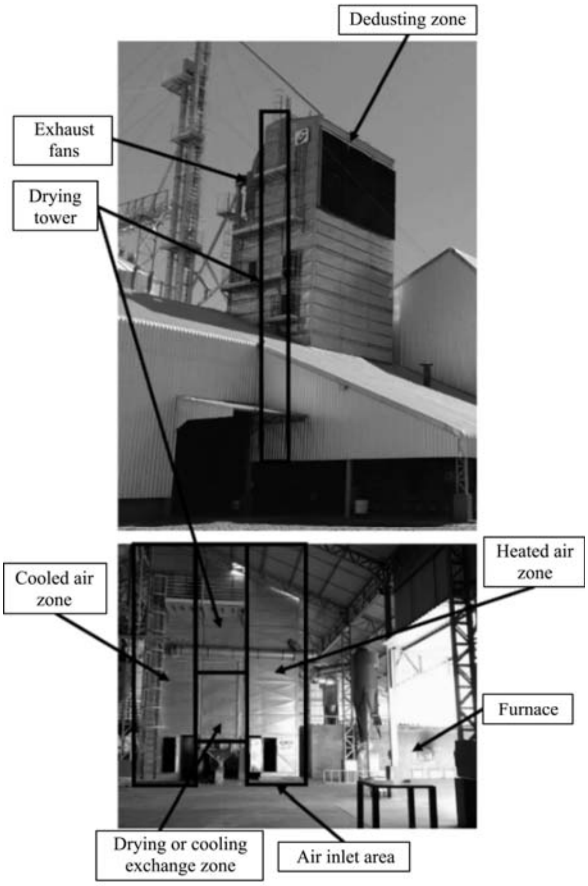


Figure 1 Schematic of the mixed-flow dryer and its components

The experimental development was focused on the definition of the velocity contour at the inlet of the dryer that was collected inside the dryer since the operation was performed with the fixed bed dryer, packed soybean and air at room temperature with the flow rate of 180 000 m^3/h. To collect the velocities that made up the inlet velocity contour of the dryer, a digital anemometer made by Minipa, model MDA-11 was used. The anemometer has a working range for temperature from 0°C to 60°C, with a reading range of speed from 0 to 30 m/s, with a resolution of 0.01 m/s and 3% full-scale accuracy. Mapping of the inlet dryer speeds was done by dividing the inlet zone area into a 14×3 grade defining its velocity collection points as a function of the length and height of the section area by setting up a table of reference for later filling with the collected data. The inlet region and grid proposed are shown in Figure 2.

Figure 2 Inlet region area of dryer and grid proposed

For computational development, each step of the CFD modeling structure was evaluated to construct a computational model that could reproduce the fluid flow phenomena in the industrial dryer used in the study. For that, the ANSYS-Fluent software was used, and the model was developed according to the Fluid Flow-Fluent computer flow solution package[20].

The geometry was defined by the industrial dryer structure studied in this work, disregarding the dimensions and non-essential elements, defining the inlet and outlet conditions as well the walls and internal conditions, such as the tower of the dryer.

The modeling and simulation were based on a general solution for a steady-state model, based on the calculation of pressure and absolute velocities without the interaction of gravity. The fluid is the air considered as the ideal gas and the grain bed inside the tower was treated as a porous medium under laminar flow[19]. The turbulent model *k-ω* SST was used to describe the air behavior in the dryer inlet and outlet zones[2]. The solution method approached was the second-order SIMPLE Upwind[20].

The governing equations for the model applied are described below:

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{1}$$

Momentum conservation equation:

$$\frac{\partial}{\partial t}(\rho \vec{V}) + \nabla \cdot (\rho \vec{V}\vec{V}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \vec{F} \tag{2}$$

where, $\vec{F}$ represents the applied external forces and the tension tensor $\bar{\bar{\tau}}$ is defined as the stress tensor.

Blake-Kozeny equation for the porous medium:

$$\frac{|\Delta P|}{L} = -\frac{1}{\dfrac{D_P^2}{150}\dfrac{\theta^3}{(1-\theta)^2}}\mu v_s \tag{3}$$

where, the medium particle diameter (D_P) is 7 mm and soybean porosity (θ) is 0.361.

Transport equations for the *k-ω* SST model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial X_i}(\rho k u_i) = \frac{\partial}{\partial X_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial X_j}\right] + G_k - Y_k + S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial X_j}(\rho \omega u_j) = \frac{\partial}{\partial X_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial X_j}\right] + G_\omega - Y_\omega + D_\omega + S_\omega \tag{5}$$

The turbulent viscosity (μ_t) is given by:

$$\mu_t = \frac{\rho k}{\omega}\frac{1}{\max\left[\dfrac{1}{\alpha^*}, \dfrac{SF_2}{\alpha_1 \omega}\right]} \tag{6}$$

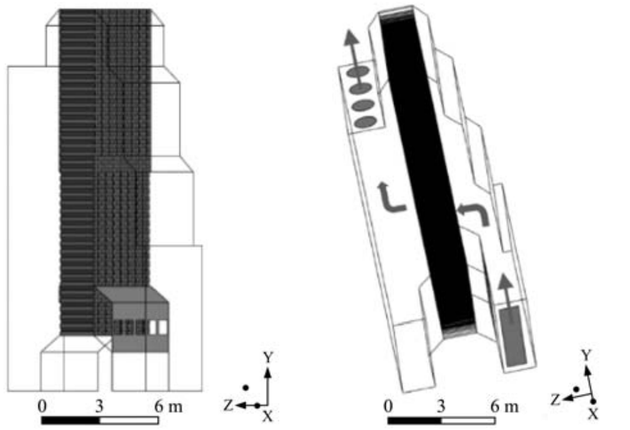
in which,

$$F_2 = tangh\left\{\left(\max\left[2\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega}\right]\right)^2\right\} \tag{7}$$

where, G_κ represents the generation of turbulent kinetic energy for the average velocity gradient; G_ω represents the generation of ω; the terms σ_k and σ_ω are Prandtl's turbulent constants for k and ω respectively; Y_k and Y_ω represent the dissipation of k and ω during turbulence; D_ω represents the cross-diffusion term; μ_t is the turbulent viscosity; y is the distance to the next surface; S is the module of the mean strain tensor and S_κ ande S_ω are user-defined constants[20].

3 Results and discussion

To develop the work, the geometry of the dryer was obtained, disregarding the dimensions and elements that were not essential to the study of the domain, defining the porous and fluid zones of the dryer. Also, the inlet and outlet boundary conditions were obtained and the dryer tower as a wall condition (Figure 3).



a. Wall boundary condition   b. Inlet and outlet boundary conditions with flow of air

Figure 3 Details of tower contour conditions, dryer inlet and outlet

The mesh presents more than 90% of tetrahedral elements, where the remaining 10% are composed of hexahedral and pyramidal elements. The number of elements and other mesh quality information is presented in Table 1.

Table 1 Quality parameters obtained for the control mesh.

Mesh quality parameters			
Quantity of elements	6 357 158		
Quality parameters	Elements quality	Orthogonal quality	Skewness
Minimum	0.19250	0.10782	0.00004
Maximum	0.99999	0.99908	0.89218
Mean	0.83044	0.76401	0.23977
Standard deviation	0.10373	0.12856	0.13343

For the simulations, a computer running Windows 10 Enterprise 2016 LTSB version 1607 was used. The processor is an Intel® Core™ i5-4210U CPU @ 1.70GHz 2.4 GHz, with 16 GB of installed RAM, 64-bit operating system and ×64-based processor. The simulations were performed with the dedicated machine using 100% processor and memory capacity.

The adopted *k-ω* SST model showed convergence for all parameters with residuals below 3×10^{-3} (continuity, axis velocity, k and ω) after 650 iterations with a difference in the inlet and outlet mass flow in order of 0.0023 kg/s. The results obtained for the convergence of the simulation indicate that the *k-ω* SST model has low computational effort (simulation time to achieve convergence).

For the analysis of the dryer pressures, we can observe the following values and deviations in relation to the working pressures, according to Table 2.

Table 2 Simulated pressure compared with working pressure for degrain dryer

Simulated pressure results vs operational data pressure:			
Pressure	Working pressures	κ-ω realizable model	
		Simulation results	Error from operational pressure
Inlet pressure	−176.40 Pa	−181 Pa	2.6%
Mixed zone pressure	−215.60 Pa	−186 Pa	13.7%
Outlet pressure	−294.00 Pa	−283 Pa	3.7%
Porous zone ΔP	−117.60 Pa	−102 Pa	13.3%

It is important to note that the pressure errors found are acceptable values for the inlet and outlet pressures while they are slightly higher for the mixed zone pressure and porous zone ΔP, which may be mainly due to the approximations made by the simplification of geometry in these sectors of the dryer.

Using the configuration designed for experimental data collection, the velocity values for the dryer inlet were extracted from the simulation at the same contour points (nodes) as the experimental data, obtaining the velocity contour for the dryer inlet according to Figures 4 and 5.

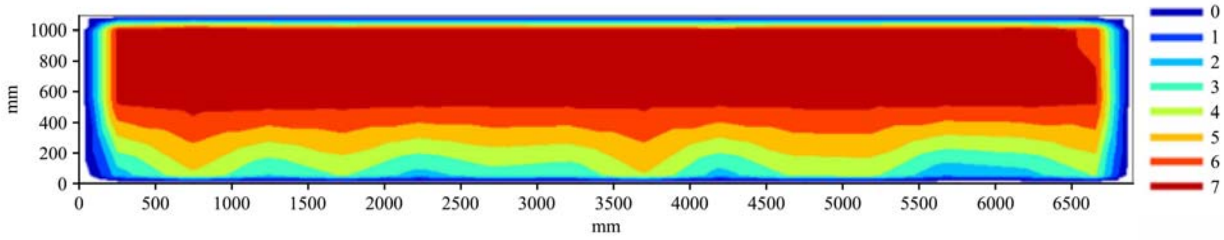


Figure 4 Contour graph of dryer inlet velocity – k-ω SST model

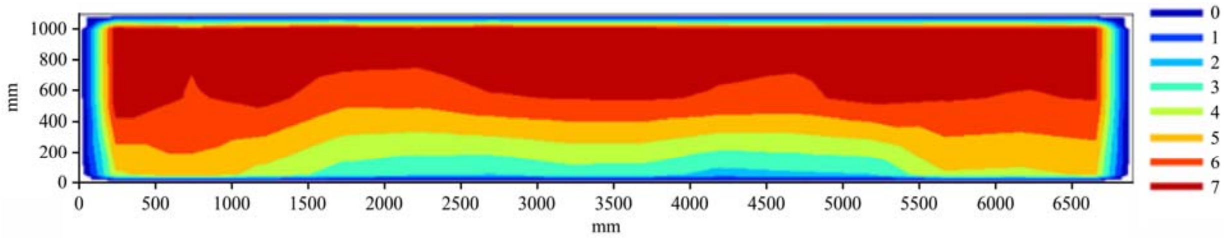


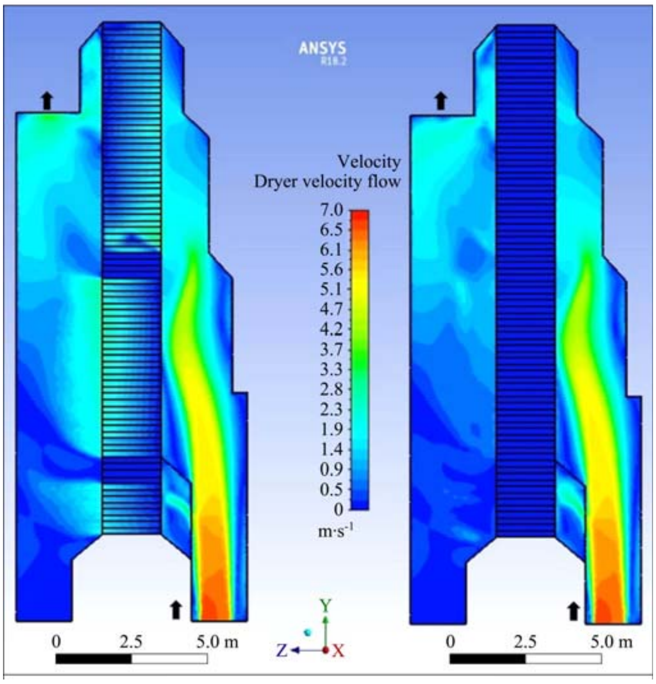
Figure 5 Contour graph of dryer inlet velocity – experimental data

The model and the experimental data present the same behavior observed in the contour graphs (Figures 4 and 5). Speeds tend to be slightly higher as it advances to the drying tower side. The velocities in the central position on the horizontal axis of measurements representing the side near the tower and the middle of the air inlet section area in the dryer respectively show lower deviation and absolute error between the model compared to the experimental data, while the position below the horizontal axis that represents the furthest entry from the drying tower shows a higher deviation and error between the model and experimental data, representing a deviation of the model from the experimental data. Differences for measurements made on the horizontal axis in the center position ranged in absolute velocity values from 0.04 to 0.9 m/s, representing a maximum deviation of 15% in the measured values. The average inlet speed of the dryer was 6.57 m/s for the model. A value was close enough to validate the simulated model when compared to the velocity value for a work flow rate of 50 m³/s at the dryer inlet speed of 6.61 m/s.

The contour profiles created by the velocity data obtained for the proposed mesh in both the simulated model and the experimentally obtained data present similar velocity contours, where the contours present higher velocities in the section near the drying tower falling along with the distancing profile from the tower. It is safe to say that the model is sufficient to predict the velocity behavior at the dryer inlet which represents that the simulation is able to predict the velocities in the entire dryer vector field in its contours.

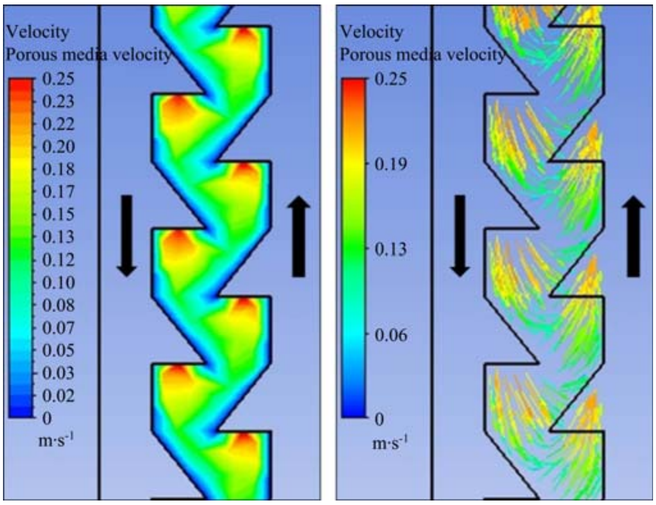
The bed and walls of the dryer have the velocity decreases tending to zero. Behavior was consistent with physically expected fluid flow due to the interaction of viscous forces in the fluid zone. Also important is the dryer air exhaust regime which corroborates the fluid dynamics of the dryer fluid zone with higher velocity in the central and upper dryer regions as the pressure drop imposed by the fixed soybean bed forcing air suction (exhaustion) through this region. In Figure 6 we can see the behavior where the velocities present higher values around 7 m/s in the regions near the drying tower with a tendency of reduction when it moves away from the tower. One factor that influences this velocity behavior is due to the dryer geometry of the position of the exhausted fans and the dryer taper in the air inlet as its height is increased. In Figure 6b, we can see the velocity drop inside the porous zone.

The velocity inside the tower had no values higher than 2.5 m/s for the fluid zone, where its average value was in the order of 1.25 m/s reaching values close to zero in the dead air flow zones by the tower according to the geometry of the dryer. The velocities in the porous medium within the tower oscillated between 0 and 0.26 m/s as shown in Figure 7 this behavior reproduces throughout the tower regions excluding dead air flow zones according to geometry dryer that was close to zero. Figure 7 presents details of contours and velocity vectors for the porous medium. It is clear the mixed-flow behavior inside the grain tower (porous medium).



a. Velocity contour passing through tower's fluid zone b. Velocity contour passing through the tower's porous zone

Figure 6 Velocity contour inside the dryer



a. Contour b. Vector velocity in the porous medium

Figure 7 Velocity inside the porous medium

4 Conclusions

The present study evaluated the fluid dynamics in a mixed-flow grain dryer in a fixed bed regime, operating in a permanent and isothermal regime using CFD techniques. The simulation results performed with the *k-ω* SST turbulence model were compared with pressure operational and velocity experimental data for model validation, where:

1) Compared to operating pressures, simulated pressures showed absolute error less than 4% for dryer inlet and outlet and absolute error less than 15% for tower mixed zone and ΔP pressures;

2) The average inlet velocity of the dryer for the simulation and operational data, presented very close values being 6.57 m/s for the simulated model and 6.61 m/s for operational data representing an absolute error of 0.6%;

3) The simulated and experimental dryer inlet velocities presented an absolute error of 15% for the inlet boundary condition in the higher velocity regions away from the influence of the inlet section walls;

4) The velocity contours at the dryer inlet show similar behavior both in simulation and experimental data.

With this finding, it can be concluded that the main objective of this work was achieved since the computable CFD method for turbulent *k-ω* SST model with porous media application can be applied in the simulations and designs of industrial mixed-flow dryers operating in a fixed bed for fluid flow evaluation.

Acknowledgements

The authors acknowledge that the CAPES from the Education Ministry of Brazil (MEC) for financial support and would like to specially thank Mr. Aparecido Visconcini for support and allowing us to use the industrial fixed bed mixed-flow grain dryer to develop our experimental data for validation of our fluid flow simulation.

[References]

[1] Brooker D B, Bakker-Arkema F W, Hall C W. Drying and storage of grains and oilseeds. New York: Van Nostrand Reinhold, 1992; 462p.

[2] Scaar H, Franke G, Weigler F Delele M, Tsotsas E, Mellmann J. Experimental and numerical study of the airflow distribution in mixed-flow grain dryers. Drying Technology, 2016; 34(5): 595–607.

[3] Bruce D M. Simulation of multiple-bed concurrent-, counter-, and mixed-flow grain dryers. Journal of Agricultural Engineering Research, 1984; 30(4): 361–372.

[4] Courtois F, Abud-Archila M, Bonazi C, Meot J, Trystram G. Modeling and control of a mixed-flow rice dryer with emphasis on breakage quality. Journal of Food Engineering, 2001; 49: 303–309.

[5] Kowalski S J, Pawlowski A. Modeling of kinetics in stationary and intermittent drying. Drying Technology, 2010; 28: 1023–1031.

[6] Khatchatourian O A, Vielmo H A, Bortolaia L A. Modelling and simulation of cross flow grain dryers. Biosystems Engineering, 2013; 116: 335–345.

[7] Cao C W, Yang D Y, Liu Q. Research on modeling and simulation of mixed flow grain dryer. Drying Technology, 2007; 25(4): 681–687.

[8] Matthies H J. Der Strömungswiderstand beim Belüften Landwirtschaftlicher Erntegüter; Düsseldorf, Germany: VDI-Verlag. 1956; 454p.

[9] Brooker D B. Pressure patterns in grain-drying systems established by numerical methods. Transactions of the ASAE, 1961; 4(1): 72–74.

[10] Thorpe G R, Hunter A J. Expressions for pressure and velocity distributions in non-uniformly aerated grain bulks. Journal of Agricultural Engineering Research, 1977; 22(1): 27–35.

[11] Smith E A. 3-dimensional analysis of air velocity and pressure in beds of grain and hay. Journal of Agricultural Engineering Research, 1982; 27(2): 101–117.

[12] Hunter A J. Pressure difference across an aerated seed bulk for some common duct and store cross-sections. Journal of Agricultural Engineering Research, 1983; 23(5): 437–450.

[13] Olesen H T. Grain Drying. Innovation development engineering, Aps. Aasvej 21, 7700 Thisted, Denmark, 1987; 161p.

[14] Cenkowski S, Miketinac M, Kelm A. Airflow patterns in a mixed-flow dryer. Journal of Canadian Agricultural Engineering, 1990; 32(1): 85–90.

[15] Sun L, Arnauld G, Fohr J P. Air flow in a corn drier. In Proceedings of the 7th International Drying Symposium (Drying '91), Prague, Czechoslovakia: Elsevier Science Ltd, 1991; pp.447–454.

[16] Roman F, Stahl-Schafer V, Hensel O. Improvement of air distribution in a fixed bed dryer using computational fluid dynamics. Biosystems Engineering, 2012; 112(4): 359–369.

[17] Scaar H, Weigler F, Mellmann J. Numerical simulation of airflow distribution in mixed-flow dryer. Acta Horticulturae, 2013; 1008: 113–118.

[18] Rupesh K R, Jyeshtharaj B J. CFD modeling of pressure drop and drag coefficient in fixed and expanded beds. Elsevier, chemical engineering research and design, 2008; 86: 444–453.

[19] Prukwarun W, Khumchoo W, Seancotr W, Phupaichitkun S. CFD simulation of fixed bed dryer by using porous media concepts: Unpeeled longan case. Int J Agric & Biol Eng, 2013; 6(1): 100–110.

[20] ANSYS INC. ANSYS Fluent–Theory guide release 19.1, 2018; 814p.