AIRFLOW PATTERNS IN A LABORATORY BATCH-TYPE, TRAY AIR DRYER

Dimitrios A. Tzempelikos¹, Alexandros P. Vouros², Achilleas V. Bardakas², Andronikos E. Filios² and Dionissios P. Margaris¹

¹ Fluid Mechanics Laboratory, Department of Mechanical Engineering and Aeronautics, University of Patras, GR-26500 Patras, Greece. e-mail: margaris@mech.upatras.gr

² Laboratory of Fluid Mechanics and Turbomachinery, Department of Mechanical Engineering Educators, School of Pedagogical and Technological Education (ASPETE), GR-141 21 Athens, Greece. e-mail: aefilios@meed-aspete.net

Keywords: Batch dryer, Airflow, CFD, Simulation.

Abstract. The batch-type dryer is one of the most popular equipment for fruit drying. However, the optimization of the air distribution inside the drying chamber remains a very important issue, due to its strong impact on efficient drying and uniformity of moisture content of products. A new scale laboratory batch-type, tray air dryer has been designed, constructed and evaluated for drying several horticultural and agricultural products. The flow field inside the dryer is studied through computational fluid dynamics (CFD). A three-dimensional model for laboratory dryer has been created and the steady state incompressible, Reynolds-Averaged Navier-Stokes (RANS) equations that formulate the flow problem are solved, incorporating standard and RNG k-ε turbulence models. The simulations are conducted for testing chamber average velocity of 2.9 m/s at ambient temperature. The CFD models are evaluated by comparing airflow patterns and velocity distributions to measured data. Numerical simulations and measurements convince that the new scale laboratory batch-type dryer is able to produce a sufficiently uniform air distribution throughout the testing chamber of the dryer.

1 INTRODUCTION

One of the most important factors in designing conventional batch-type air dryers is the airflow design. In industrial air dryers the effect of flow heterogeneity is particularly difficult to resolve. The distribution of airflow depends on the process of drying, the drying medium, the geometry and the equipment of the drying chamber. These factors determine the uniformity of drying and quality of finished products. Even though the performance of a drying chamber can be studied experimentally, the time and cost limits of research restricts the generalization of the results and certainly cannot be applied to the original design of the drying chamber. In contrast, with the help of computational fluid dynamics (CFD), spanning a wide range of industrial and non industrial applications, the complexity of the flow field can be solved numerically.

Mathioulakis et al [1] simulate the flow of air in an industrial batch-type, tray air dryer. The distribution of pressure and velocity over the product were found to have a lack of spatial homogeneity leads to variations in drying rates and moisture contents. Margaris and Ghiaus [2] simulated the airflow in an industrial drier and provided parameters for different configuration helped to optimize the drying space with significant improvement in the quality of the dried product and the reduction of energy consumption. Miracle [3] used a two-dimensional CFD model with time dependent boundary conditions, studying the distribution uniformity of air velocity in an industrial dryer meat for low and high levels of ventilation cycle. Hoang et al [4] simulated the airflow into a cold store solving the steady state incompressible, Reynolds-averaged Navier-Stokes (RANS) equations, applying the standard k-ε and the RNG k-ε turbulence models. The results showed that the RNG k-ε model does not help to improve the prediction of recirculation and any improvement requires finer grid with enhanced simulation of turbulent flow. Amanlou and Zomordian [5] designed a new fruit cabinet with different geometries and simulated them using CFD. The experimental results and the predicted data from CFD revealed a very good correlation coefficient for drying air temperature and air velocity in the drying chamber. Norton and Sun [6] in a review paper show the widely use of CFD for predicting the air velocity and temperature in drying chambers and Scott and Richardson[7] and Xia and Sun[8] present the commercial CFD packages that have been increasingly deployed in the food industry.

Recent studies show that limited research on the prediction and measurements of flow and pressure fields in batch-type air dryers has been performed. The absence of experiments can be attributed to the difficulty of direct measurement of local air velocity and flow in a drying chamber of horticultural and agricultural products.
The present study concerns the design, construction and evaluation of a new scale laboratory batch-type, tray air dryer which can serve thermal drying studies in fully controllable environment. The velocity and pressure fields are analyzed with the aid of CFD commercial code Fluent®. For the numerical simulations the steady state RANS equations are solved in combination with the standard k-ε and the RNG k-ε turbulence models. The effect of the turbulence modeling is distinguished through direct comparisons of the derived airflow patterns. The purposes of the current research are: a) the study of the velocity fields in the drying chamber of new scale laboratory batch-type, tray air dryer building a CFD method that is affordable in terms of computation time and b) the comparison of the numerical outcomes with experimental measurements performed with a sensor velocity.

2 EXPERIMENTAL SETUP AND MEASUREMENTS

2.1 Description of the dryer

The lab scale batch-type, tray air dryer which has been designed and constructed in the Laboratory of Fluid Mechanics and Turbomachinery in ASPETE, is shown in Figures 1 and 2. The overall dimensions of the facility are 4,7 m (length), 2,5 m (width) and 2,5 m (height). The construction of the air ducts are made from steel of 0,8 mm thickness. All the ducts are insulated with 10 mm Alveolen (Frelen) with thermal conductivity of 0,032 W/mK and water absorption of 0,011 kg/m².

![Figure 1. Schematic diagram of the lab-scale batch-type, tray air dryer](image)

![Figure 2. Photo of the lab-scale air dryer, equipped with measuring instrumentation and data acquisition system](image)

The square section drying chamber (0,5m x 0,5m) is of tower (vertical) type and is equipped with a metal tray which is supported on four, side walls mounted, load cells. A set of four refractory glasses 10 mm thickness are available to replace the side steel walls when optical clarity and precise visual observations are required.

Upstream of the drying chamber, the long rectangular diffuser with total divergence angle 6,7°, the tube heat exchanger in which the hot water is provided through a boiler of 58 kW (50.000 kcal/h) thermal power, the transitional duct with observation window that includes the sprayer for humidifying purposes, the corner duct that incorporates four guide vanes and the flow straighteners section, are located. The flow straighteners consisting from an aluminum honeycomb (made form 3003 aluminum alloy foil) with cell size of 1/4' and 38 mm thickness and screen wires located downstream of the honeycomb, are considered necessary aiming to the
Dimitrios A. Tzempelikos, Alexandros P. Vouros, Achilleas V. Bardakas, Andronikos E. Filios and Dionissios P. Margaris.

Flow uniformity in the drying section. The flow rate is observed and controlled with a custom made and calibrated rake of pitot tubes, namely pitot rake, located at the inlet of the drying chamber. Downstream of the vertical drying chamber, the second corner duct with guide vanes, the elevated horizontal modular constructed duct, the outlet dumper and the exit diffuser, are located. The modular design of the facility permits the easy place of two or three horizontal drying chambers in tandem arrangement, on the elevated return or exit flow leg.

The air flow is established and controlled through a centrifugal fan directly driven by a 3 phase electric motor of 3 kW with its speed regulated by an ac inverter. Adjusting the air dampers, the laboratory dryer would be used for thermal drying experimental studies in both open circuit and close circuit operations.

2.2 Measurements

Air velocity surveys into the drying chamber under ambient conditions, i.e. atmospheric pressure at 18.4 °C were carried out for constant speed the induced centrifugal fan at 690 rpm in 23 Hz. The volumetric flow rate was 2.600 m³/h, resulting to a mean velocity 2.9 m/s and Reynolds number 9.9x10⁴ (based on hydraulic diameter of the drying chamber).

The mean speed of the air flow at the inlet is the weighted average velocity of the 12 points collected from the pitot rake arrangement as shown in Figure 3 and the four pressure taps (same level with the contact tip of the pitot tube) in the side wall of the inlet of the drying chamber.

Each pitot is connected via plastic tubing to a custom made pressure collector system equipped with solenoid valves (Tekmatic 24VDC, 6W) permitting its operation and control with the use of custom software developed in Labview®. A differential pressure transmitter (Dwyer, model MS-121-LCD) with a calibrated accuracy ± 2% of the selected range of 25 Pa was used to measure each of the 12 points with automatically “open-close” the proper solenoid valve.

Aiming to the cross check of the velocities measured by pitot tubes and static pressure holes, a velocity reference transducer (54T29, Dantec Dynamics® with 54N81Multichannel CTA) was used which is the best balance between cost and accuracy. The velocity range of the sensor is 0 – 30 m/s. The calibrated accuracy is ± 2% of reading ± 0.02 m/s or ± 2.6 % of the selected range of 3 m/s, which is assured by a certificate provided by the manufacturer.

The measurement of the velocity was done inside the duct at distance 510 mm from the inlet of the drying chamber. For measuring the air velocity during each test, at different locations of the drying chamber, 4 holes on the side wall of the drying chamber were pierced (Figures 2 and 4). All holes except the one through which velocity transducer was inserted for air velocity measurement, were filled tightly by conic plastic washers. Inlet air velocity was kept constant during the experiment.

For reading velocity at each point inside the drying cabinet, the velocity transducer was inserted through a side wall proximity hole and adjusted at eight different locations along the depth of the drying chamber. At each point the time averaged velocity was determined from measurement with frequency of 200 Hz (200 samples/sec) and averaged over a 10 sec period. The experimental values are directly compared with the numerical predictions at the same locations.

Both the differential pressure transmitter and the velocity transducer are connected to a PC with a PCI-e-6321 DAQ device (National Instruments®) via NI SCXI-1000 and NI SCXI-1302 modules. Custom software through Labview® was used to interface with the data acquisition.

The overall accuracy of the CFD calculations is calculated as the average of the absolute differences between the time-averaged velocity magnitude for the CFD calculation and the measurement at each position, divided by
the average velocity magnitude in the drying chamber obtained from the measurements and is expressed as:

\[ E = \frac{\sum_{i=1}^{n} |U_{\text{exp}}^{i} - U_{\text{cfd}}^{i}|}{\sum_{i=1}^{n} U_{\text{exp}}^{i}} \times 100 \]  

(1)

where \( U_{\text{cfd}}^{i} \) is the average velocity at position \( i \) from the CFD calculation, \( U_{\text{exp}}^{i} \) is the average velocity at position \( i \) from the measurement and \( n \) is the number of measurement points.

3 NUMERICAL SIMULATION

Numerical computation of fluid transport includes conservation of mass, momentum and turbulence model equations. The preprocessor Gambit® was used to create geometry, discretize the fluid domain into small cells to form a volume mesh and set up the appropriate boundary conditions. The flow properties were then specified, the equations were solved and the results were analyzed by Fluent®.

3.1. Governing equations

The governing equations based on the conservation of mass and momentum of a newtonian fluid flow which applied to an infinitesimal small volume in a cartesian co-ordinate system (x, y, z) and using the Reynolds averaged formulation [9], are:

\[ \frac{\partial \rho}{\partial t} + \text{div} \mathbf{U} = 0 \]  

(2)

\[ \frac{\partial (\rho u_i)}{\partial t} + \text{div} (\rho \mathbf{U} u_i) = \text{div} (\mu_{\text{eff}} \text{grad} u_i) - \frac{\partial p}{\partial x_i} + S_i \]  

(3)

\[ \rho = \rho(p, T) \]  

(4)

\[ \mu_{\text{eff}} = \mu + \mu_T \]  

(5)

In these formulae, \( \mathbf{U} \) is the velocity vector, consisting of three components \( u_x, u_y, u_z \) (m/s), \( p \) is the pressure (Pa), \( T \) is temperature (°C). The density \( \rho \) (kg/m³) and the laminar viscosity \( \mu \) (kg/ms) are the only fluid properties involved; \( \mu_T \) and \( \mu_{\text{eff}} \) are the turbulent and effective viscosity, respectively. The sources \( S_i \) contain further contributions of the viscous stress term and can contain additional body forces. The airflow at a constant temperature in the air dryer is considered (\( \rho = 1,225 \text{ kg/m}^3 \) and \( \mu = 1,7894 \times 10^{-5} \text{ Ns/m}^2 \)).

3.2. Turbulence models

The k-\( \varepsilon \) models are the most widely validated turbulence models in literature and are the standard models to use in the commercial codes. The k-\( \varepsilon \) model uses an eddy-viscosity hypothesis for the turbulence, expressing the turbulent stresses as an additional viscous stress term [Eq. (5)]. In the k-\( \varepsilon \) model, the turbulent viscosity is expressed in terms of two variables: the turbulence kinetic energy \( k \) and its dissipation rate \( \varepsilon \). In this model, the \( k \) and \( \varepsilon \) equations are of a similar conservation format, containing convective and diffusive terms.

3.2.1. The standard k-\( \varepsilon \) model

In the standard k-\( \varepsilon \) model, the production and destruction terms in the \( \varepsilon \) equation contain empirical constants. The resulting equations are similar to the governing flow equations [9]:

\[ \mu_T = C_\mu \frac{k^2}{\varepsilon} \]  

(6)

\[ \frac{\partial k}{\partial t} + \text{div} (\rho \mathbf{U} k) - \text{div} \left[ \mu + \frac{\mu_T}{\sigma_k} \right] \text{grad} (k) = P - \rho \varepsilon \]  

(7)

\[ \frac{\partial \varepsilon}{\partial t} + \text{div} (\rho \mathbf{U} \varepsilon) - \text{div} \left[ \mu + \frac{\mu_T}{\sigma_\varepsilon} \right] \text{grad} (\varepsilon) = C_{\mu k} \frac{\varepsilon}{k} P - C_{2\varepsilon} \frac{\varepsilon^2}{k} \]  

(8)

where \( P \) is a term containing the turbulence production due to the stresses in the flow. The standard k-\( \varepsilon \) model contains five empirical constants (\( C_\mu, C_{1k}, C_{2\varepsilon}, \sigma_k \) and \( \sigma_\varepsilon \)) and during this study these constants were not
changed:

\[ C_\mu = 0.09, \quad C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad \sigma_k = 1.0 \text{ and } \sigma_\varepsilon = 1.3 \quad (9) \]

Near walls the equations do not hold and standard logarithmic wall profiles are implemented. An important variable is \( y^+ \), a dimensionless distance normal to the wall. The value of \( y^+ \) determines in which region of the boundary layer the first node is situated. The log-law is valid only for \( y^+ > 30 \).

### 3.2.2. RNG k-\( \varepsilon \) model

The RNG k-\( \varepsilon \) model is obtained via a statistical mechanics approach, in which small scale motions are systematically removed from the governing equations by expressing their effects in terms of larger scale motions and a modified viscosity. Furthermore, a new term appears in the \( \varepsilon \)-equation, which account for anisotropy in strongly strained turbulent flows. This term is incorporated through a modeled constant in the production term, based on the equilibrium assumption that production equals dissipation, restricting the RNG k-\( \varepsilon \) model to a coarse grid approach near walls. The same default wall functions as in the standard k-\( \varepsilon \) model are valid in this case. The \( \varepsilon \) equation is \([9]\):

\[
\frac{\partial \varepsilon}{\partial t} + \text{div}(\rho U \varepsilon) - \text{div} \left( \mu + \frac{\mu_k}{\sigma_\varepsilon} \right) \text{grad}(\varepsilon) = (C_{1\varepsilon} - C_{\text{RNG}}) \frac{\varepsilon}{k} P - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (10)
\]

\[
C_{\text{RNG}} = \left( \frac{1 - \frac{n}{n_0}}{1 + \beta n^+} \right) \quad (11)
\]

\[
n = \left( \frac{P_s}{\mu_T} \right)^{0.5} \frac{k}{\varepsilon} \quad (12)
\]

where \( n_0 \) and \( \beta \) are additional model constants, which are equal to 4.38 and 0.012 respectively and \( P_s \) is the shear part of the production. The standard values of the other constants are regarded as suitable for this application:

\[ C_\mu = 0.0845, \quad C_{1\varepsilon} = 1.42, \quad C_{2\varepsilon} = 1.68, \quad \sigma_k = 0.7179 \text{ and } \sigma_\varepsilon = 0.7179 \quad (13) \]

The k equation is the same format as in the standard k-\( \varepsilon \) model and the same wall profiles are applied.

### 3.3. Model of the tray

A source term was added to k-\( \varepsilon \) and RNG k-\( \varepsilon \) turbulence models equations to estimate the pressure drop across the tray into the drying chamber. The tray was calculated as a screen and in the CFD simulation, the screen was modeled as a thin porous media of finite thickness over which the pressure change was defined as a combination of Darcy’s Law and an additional inertial loss term and is given by \([10]\):

\[
\Delta p = - \left( \frac{\mu}{\alpha} U + C_2 \frac{1}{2} \rho U^2 \right) \Delta m \quad (14)
\]

where \( \mu \) is the laminar fluid viscosity, \( \alpha \) is the permeability of the medium, \( C_2 \) is the pressure-jump coefficient, \( U \) is the velocity normal to the porous face, and \( \Delta m \) is the thickness of the medium.

### 3.4. Model of the air dryer

The flow field into the drying chamber of empty laboratory batch-type, tray air dryer, operated in open circuit mode is numerically studied. The structure of the modeled dryer is depicted in Figure 5. The dryer is 4.7 m in length, 0.5 m in width and 1.38 m in height. The dimensions of the drying chamber are 0.5 x 0.5 x 0.66 m. The air dryer is modeled with the tray located in a distance 0.29 m from the inlet of the drying chamber. The tray has a length of 0.48 m, width of 0.48 m and thickness 2.8 mm. The shape of the tray has been modeled as a screen with orthogonal holes. The dimensions of the orthogonal holes are 23.74mm x 10.9mm. A gap of 10 mm existed between the tray and the wall of the drying chamber.
The velocity profile at the entrance of the drying chamber was measured by 12 points (Figure 3). These points were used as an inlet boundary condition for the simulation performed and for that reason the geometry of the fan, the diffuser, the tube heat exchanger, the down guide vanes and the flow straighteners are not modeled.

3.5. Description of the numerical simulation

The calculations have been performed with Fluent®. In the steady RANS simulations of the airflow into the batch-type tray air dryer, the standard k-ε and the renormalization group RNG k-ε turbulence models have been used. The standard k-ε model which valid only for fully turbulent flows is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for the turbulence kinetic energy is derived from the exact equation, while the model transport equation for the dissipation rate is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k-ε model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The RNG k-ε model employs a differential form of the relation for the effective viscosity, yielding an accurate description of how the effective turbulent transport varies with the effective Reynolds number. This allows accurate extension of the model to near-wall flows and low-Reynolds-number or transitional flows. The standard logarithmic wall functions, which are a collection of semi-empirical formulas and functions were applied to bridge the viscosity affected region between the wall and the fully turbulent region. The SIMPLE algorithm has been used together with the solver of Fluent® to solve the pressure-velocity coupling equations. In order to improve numerical accuracy, the second-order-upwind scheme has been used to discretize the RANS equations.

3.6. Numerical solution control

For the numerical simulations, a desktop PC (Intel® Core i7 CPU at 2,67 GHz) was used. The number of iterations has been adjusted to reduce the scaled residual below the value of 10^{-5} which is the criteria. For each run, the observation of the integrated quantities of total pressure, at suction as well as at discharge surface was appointed for the convergence of the solution. In many cases this drives the residuals in lower values than the initially set value. Depending on the case, the convergence was achieved at difference iterations, as the result at a specific mass-flow was used to initialize the computations at another mass-flow. Aiming to smooth convergence, various runs were attempted by varying the under-relaxations factors. In that way, a direct control regarding the update of computed variables through iterations was achieved. Initializing with low values for the first iterations steps and observing the progress of the residuals, their values were modified aiming to the acceleration of the convergence.

3.7. Boundary conditions and mesh cells

The inlet boundary condition of the model was set as an inlet velocity profile by using a set of velocity measured at 12 points inside the air dryer (Fig 3). The direction of the velocity was normal to the inlet boundary and the average velocity was 2,89 m/s. The turbulent intensity, which is defined as the ratio of the root-mean-square of the velocity fluctuations, to the mean flow velocity can be estimated from the following formula derived from an empirical correlation for pipe flows [10],

\[ I = \frac{u'}{u_{avg}} = 0,16(Re_{th})^{-1/8} \]

(15)

The turbulent intensity at the inlet boundary was set as 3,8%, based on Re=9,9x10^4. An atmospheric pressure boundary located downstream of the outlet duct was specified as pressure outlet. The no slip boundary condition was used in all the walls. Porous jump boundary condition was used for the tray and appropriate values for...
The CFD model of new scale laboratory batch-type dryer consists of about $8 \times 10^5$ computational nodes (Figure 6). A body fitted structured grid was used. The grid was refined close to the walls, in between the wall and the tray and inside the tray. The $y^+$ was in the range of 30 to 50. The grid independence was checked and a converged solution was obtained after approximately 1900 iterations.

4 RESULTS AND DISCUSSIONS

The predicted velocities with k-$\varepsilon$ and RNG k-$\varepsilon$ at height $z=0.51$m, as shown in Figure 4, are compared with the corresponding experimental values. Figure 7 shows a comparison between the measured and the simulated values of air velocity distribution, which give an overall accuracy of the CFD calculations, $E$ in equation (1), 2.79 % for the k-$\varepsilon$ and 2.72 % for the RNG k-$\varepsilon$ turbulence model. The difference of the absolute between the simulated and experimental values varies from 0.002 to 0.227 m/s for k-$\varepsilon$ and 0.002 to 0.213 m/s for RNG k-$\varepsilon$ turbulence model. The relative error between the simulated and experimental values varies from 0.08 to 7.38 % for k-$\varepsilon$ and 0.08 to 6.93 % for RNG k-$\varepsilon$ turbulence model. The average velocity of the experimental values is 3.22 m/s with a standard deviation 0.12761. The average velocity and standard deviation for the k-$\varepsilon$ and RNG k-$\varepsilon$ turbulence model is 3.274 m/s, 0.09367, 3.267 m/s and 0.10503 respectively. The overall accuracy of the CFD calculations indicates the CFD simulation scheme is practical for the analysis of the velocity field in the drying chamber. Figure 8 illustrates the turbulent intensity predicted with k-$\varepsilon$ and RNG k-$\varepsilon$ at position $z=0.51$m. The average turbulent intensity is about 4%. At the edges of the drying chamber (5 cm for the wall) the turbulent intensity reaches almost 14%. This difference could be explained by the presence of the tray and its geometry. In Figure 9, the velocity contours chosen for their relevance concerning the assessment of the airflow calculations are shown. It can be seen that high velocities are encountered at the center of the chamber. Near the four walls the air moves at lower velocities due to the presence of the tray which located 200 mm below the level where measurements were taken. Both the two turbulence models predict almost the same air flow distribution. Figures 8 and 9 verify that at the core of the drying chamber, the turbulent intensity of the velocity field is relatively low and the flow is homogenous.

![Figure 7. Velocity distributions at $z=0.51$m. Comparison between the measured data and CFD prediction with k-$\varepsilon$ and RNG k-$\varepsilon$ turbulence model: (a) $y=163$mm, (b) $y=223$mm, (c) $y=283$mm and (d) $y=343$mm](image-url)
Figure 8. Turbulent intensity in z = 0.51m. Comparison between numerical predictions with k-ε and RNG k-ε turbulence models: (a) y=163mm, (b) y=223mm, (c) y=283mm and (d) y=343mm

Figure 9. Velocity contours (m/s) at z = 0.51m predicted with k-ε (left) and RNG k-ε (right) turbulence models

The velocity contours at the air dryer of the developed CFD model are presented in Figure 10. The velocity contours reveal the presence of high velocities regions especially at the middle of the drying chamber and above the tray disk.

In Figure 11, the static pressure contours in the air dryer reflects the presence of low velocities regime especially at the inlet of the drying chamber and at the upper guide vanes. At a distance 310 mm from the inlet of the drying chamber, there is a pressure drop from 6 to 1 Pa in terms of gauge pressure. This drop of the static pressure is due to the presence of the tray disk at this location.

In Figure 12, δ represents the relative difference of velocity magnitude with k-ε and RNG k-ε turbulence model with respect to the k-ε turbulence model and defined as:

$$\delta = \frac{U_{k-\epsilon}^i - U_{RNG-k-\epsilon}^i}{U_{k-\epsilon}^i} \times 100$$

(16)

Near the wall of the drying chamber the parameter δ reaches almost 10% and in contrary at the middle of the chamber, the velocity predictions are independent of the turbulence model.
CONCLUDING REMARKS

A fluid flow model of a new scale laboratory batch-type, tray air dryer, including its major physical features, is developed using CFD code Fluent®. Standard k-ε and RNG k-ε turbulence models were used for computing turbulence parameters inside the air dryer. Numerically predicted velocity profiles inside the drying chamber are compared with the measured data. These predictions are found to be in reasonable agreement with measured data. The turbulence intensity is low and the homogeneity of the drying chamber is acceptable. There is a slightly difference between the k-ε and the RNG k-ε turbulence model predicting the velocity profiles and the model developed is found useful to predict the airflow pattern inside the drying chamber. Further work will focus on validating CFD results with drying experiments with organic and inorganic products in the drying chamber of the air dryer.
ACKNOWLEDGMENTS

The measuring equipment and the data acquisition of the lab batch-type dryer in the Laboratory of Fluid Mechanics and Turbomachinery has partially funded from public and private sponsors. The authors gratefully acknowledge Special Account for Research of ASPETE, Delta Pi S.A., A.A. Roibas & Co., Mr. Dionisios Tsepenakas (EKO S.A.) and Mr. Michalis Petrolekas (National Instruments, Hellas) for their kind contribution and support.

REFERENCES