AIR FLOW CHARACTERISTICS IN AN INDUSTRIAL WOOD PALLET DRYING KILN

Dimitrios A. Tzempelikos¹, Andronikos E. Filios* and Dionissios P. Margaris¹

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

* Fluid Mechanics and Turbomachinery Laboratory, Department of Mechanical Engineering Educators,
School of Pedagogical and Technological Education,
GR-141 21 Athens, Greece
e-mail: fmtulab.aspete@gmail.com

Keywords: Wood drying, airflow, cfd, pallet, kiln.

Abstract. The improvement and optimization of air-distribution systems in drying kilns contributes to the preservation of the quality, safety and shelf life of perishable products. The present study reports on the numerical solution of airflow within a two dimensional drying kiln enclosure loaded with wooden pallets. The performance of air flow field is examined with and without supply of wooden pallets. Different arrangements of the supplied wooden pallets are investigated as well as the use of a screen in order to improve the airflow filed. The steady state RANS equations that formulate the flow problem are solved along with a turbulence model that is the standard k-ε or the rng k-ε. The effect of the turbulence modeling is distinguished through direct comparisons of the derived airflow patterns. The results obtained show the presence of stagnant zones inside the pallets columns and above of them and the presence of recirculation regions in different zones of the drying kiln with the two turbulence models.

1 INTRODUCTION

Airflow design is potentially important in the design of drying kilns which operate as closed, fully recirculatory systems. In industrial wood drying kilns, the effect of non-uniform airflows is particularly difficult to resolve. The airflow distribution is dependent on the drying process, the drying medium, the geometry and equipment of the drying kiln. Although the performance of a drying kiln can be studied experimentally, the time-consuming and costly methodology restricts the generalization of the outcomes and certainly is not applicable in an early design phase of the drying chamber. In contrary, with the aid of the computational fluid dynamics (cfd) that spans in a wide range of industrial and non-industrial applications, the complex flow field can be solved numerically.

Various researchers have considerably contributed in revealing the flow mechanics inside of a closed enclosure, aiming to improve and optimize the air distribution systems. Sun et al.[1] simulating velocity and pressure distributions in an industrial dehumidifier wood drying kiln showed that for high efficiency it is important to avoid air recirculation. Smit et al.[2] simulate a two dimensional pore-scale model in order to predict air flow through a wood drying stack and the predicted results are favorably compared with experimental measurements. Langrish and Keey[3] modeling the air flow patterns in a timber kiln, intended to predict the distribution of the airflow in the fillet spaces between the boards in a hydraulic model of a timber kiln.

Margaris and Ghiaus[4] simulated extensively the air flow inside a full-scale industrial dryer and the predicted parameters for different configuration contributed in the optimization of the drying space and led to a substantial improvement of the quality of the dried product along with the reduction of energy consumption. Mureh et al.[5][9] studied the numerical and the experimental characterization of airflow within a semi-trailer enclosure loaded with pallets. The experiments were carried out on a reduced-scale model of a refrigerated air duct systems. The numerical predictions show reasonable agreement with experimental data and moreover the results saw that the supply air duct system improves significantly the homogeneity of ventilation in the semi-trailer enclosure. Mathioulakis et al.[10] simulated the air movement inside the drying chamber of an industrial batch-type, tray air dryer. The pressure and the air velocities above the product were found to have a lack of spatial homogeneity.
Canteloup and Mirade\cite{11} studied ventilation efficiency inside forced-ventilation food plants, using two transient methods and a steady-state method based on the resolution of an additional scalar transport equation which implemented and compared with experimental and numerical data. Mirade\cite{12} used a two-dimensional cfd model with time-dependent boundary conditions, estimated the homogeneity of the air velocity distribution in an industrial meat dryer for low and high levels of the ventilation cycle. Chourasia and Goswami\cite{13} studied the heat transfer and moisture loss under steady state airflow, in a commercial scale potato cold store. The proposed model could be applied to incorporate necessary design improvements aiming to the improvements of the airflow distribution that result in minimization of the storage losses. Tapsoba et al\cite{14,15} studied a reduced-scale model in order to investigate experimentally and numerically the airflow patterns within a ceiling-slot ventilated enclosure loaded by slotted boxes. The numerical predictions with the application of the Reynolds stress model show rather good agreement with experimental data. Hoang et al\cite{16} simulated the airflow inside a cold store solving the steady state incompressible, Reynolds-averaged Navier-Stokes (RANS) equations, applying the k-ε and the rng k-ε turbulence model. The results saw that rng k-ε model did not help to improve the prediction of the recirculation and any improvement requires finer grid along with enhanced turbulence modeling.

Recent investigations show that little research on the prediction and measurements of flow field in a wood drying kilns loaded with pallets has been performed. The lack of experiments can be attributed to the complexity of direct measurement of local air velocities and flow rates in the thin air spaces located between the pallets.

In this study, cfd is used to develop and solve the physico-mathematical problem that describes the steady two-dimensional flow field in a wood pallets drying kiln of industrial type that is used for the effective preservation of wood by the withdrawing of microorganisms. The performance of air flow field is examined with and without supply of wooden pallets. Different arrangements of the supplied wooden pallets are investigated as well as the use of a screen in order to improve the airflow distribution. The commercial cfd code Fluent\cite{17} is used in all numerical simulations in which the steady state RANS equations are solved in combination with the k-ε and the rng k-ε turbulence model. The effect of the turbulence modeling is distinguished through direct comparisons of the derived airflow patterns.

2 COMPUTATIONAL MODELING

2.1 Geometry of the drying kiln

The longitudinal cross section of the considered industrial type wood drying kiln is shown in Figure 1. The drying unit system consists of a dehumidifier module, a stack of pallets and one kiln air recirculation fan, all located within an insulated kiln chamber. The main elements of the dehumidifier, which influence the flow patterns in the kiln, are the condenser and evaporator. Each dehumidifier module also has an electric air heater to preheat the kiln before drying commences. For simplicity, in this investigation the dehumidifier module and the electric air heater have been incorporated with the kiln air recirculation fan.

Into the drying chamber, there are four pallets columns (i.e. PC-1 to PC-4), being 2,1 m height each in the air flow direction. The distance between the pallet columns is 36 cm. Each pallet column consists of fifteen (15) pallets equally spaced. A three dimensional view of a pallet is shown in Figure 1a and details of the horizontal board layers pallets are shown in Figure 1b. The space between the pallets is 9 cm. Each pallet in the present simulation is considered as a two dimensional rectangular solid block with dimensions 80 cm x 5 cm.

The four vertical and one horizontal dashed dot lines represents the selected sections where distributions of velocities and pressures are predicted. The vertical sections are named from 1 to 4 and horizontal section is referred as CLHS section.

2.2 Boundary conditions and mesh cells

The boundary conditions for air flow are determined by the performance of the six blades (pitch 37,5 deg) axial flow fan with an impeller diameter 872 mm. The total air volume flow rate through the stack of pallets is determined by the pressure jump across the axial fan and it results from the performance curves of the fan. At the inlet, uniform velocity distribution is assumed (U_0=16,3 m/s) that corresponds in Q_0=35,000 m³/h. At the outflow, pressure is also assumed to be uniform and zero-gradient is applied for all transport variables. The porosity of the round hole metal screen of 2 mm in thickness is 40% resulting to a resistance coefficient which is 7,4.

The turbulence models are only valid in fully turbulent regions. Close to the wall, where viscous effects become dominant, the models are used in conjunction with wall functions. For this study, the convective equilibrium logarithmic law governing the wall is used.

The computational domain is meshed structured with the aid of Fluent preprocessor Gambit\cite{17} and fine grid spacing close to all walls in order to resolve steep gradients has been applied. The final grids for the drying kiln solutions consist of 276,000 cells for the empty configuration and 227,000 cells for the loaded configuration. Both solutions are independent of the mesh size.
2.3 Description of numerical simulation procedure

The calculations have been performed with Fluent®. In the steady RANS simulations of the airflow into the drying kiln, the standard k-€ and the renormalization group (rng) k-€ turbulence models have been used. The standard k-€ 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 (€). 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-€ model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The rng k-€ 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 have been used for the near wall treatments. 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.
2.4 Numerical solution control

The code was run on an Intel® Core 2 CPU at 2.4 GHz. 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 RESULTS AND DISCUSSION

The predictions of the airflow pattern in the unloaded drying kiln are shown in Figures 2 and 3. The air coming out of the fan is accelerated, reaches the ceiling and spreads out at the upper left entrance of the drying kiln. The air flow downwards and reaches the bottom of the kiln with high velocity. From Figure 3, it can be seen that air flow recirculation eddies are produced at the top and bottom side corners of the kiln. At the center of the kiln there is a recalculation zone with low velocities. In this part of the kiln the flow is weak and velocity magnitude did not exceed 10% approximately of the $U_0$. The numerical results indicate that air flow recirculation eddy is produced at all the top and bottom corners of the kiln and the region of outlet as well. With no differences, both turbulence models, k-ε andrng k-ε, predict well the presence of the air recirculation.

![Figure 2](image1.png)

Figure 2. Vectors of velocity magnitude (m/s) predicted with k-ε (left) and rng k-ε turbulence model (right) into the empty drying kiln.

![Figure 3](image2.png)

Figure 3. Path lines of velocity magnitude (m/s) predicted with k-ε (left) and rng k-ε turbulence model (right) into the empty drying kiln.

The predictions of the airflow pattern in the loaded drying kiln are shown in Figures 4 and 5. The path lines structure which are shown in Figure 5 for both turbulence models, assure the complexity of the airflow with high velocity downstream of the kiln and air flow recirculation in side corners.

Air flow recirculation is also generated at the left side of the first pallets column (PC), above of the four series of the pallets columns and inside of them, especially at upper middle of the pallets, at the right side of the screen and finally in the outlet region. At the center of the pallets columns there are lots of recalculation zones with low
velocities. In this part of the drying kiln, the flow is weak and velocity magnitude did not exceed 40% approximately of the $U_0$. With no differences, both turbulence models, k-ε and rng k-ε, predict well the presence of the air recirculation.

In Figures 6 to 12, the spatial and flow variables are expressed in non-dimensional. The reference dimensions are: the height (H) of the chamber which is equal to 2.68 m, the length (L) of the chamber which is equal to 6.49, the mean velocity in the inlet of the chamber ($U_0$) which is equal to 16.30 m/s and the atmospheric pressure ($p_{atm}$) which is equal to 101.325 Pa.

Figures 6 to 9 (left plots) show the profiles of the lateral velocity at different vertical sections (Section 1 to 4) of the kiln. Figure 10 (left plot) show the profile of the longitudinal velocity along the centerline of the chamber. In the same figures, there is a comparison between the k-ε and the rng k-ε turbulence model in both modes regarding the application of the load. Unload means that the kiln operates empty while load means that the kiln operates with the presence of pallets.

It is seen from Figure 6, for the loaded configuration that the velocity distribution in Section 1 is not uniform at the entrance region of the first PC. The non dimensional velocity $u/U_0$ spans from 0 to 1.6 approximately. This variation in the velocity appears to be caused by non-uniform flow in the entrance of the kiln illustrated in Figure 5.

The large range in the velocity indicated that there are large recirculation eddies at the entrance of the first PC. The flow patterns of the airflow on the top entrance of the second PC are similar to those of the first PC, but the velocities are smaller. At the entrance of the third PC, the velocity magnitude is smaller than those at the entrances of the first and second PC (Figure 7).

However the large span in velocity magnitude in the exit area of the third PC indicates the presence of recirculation eddies at the exit of the PC. At the entrance of the fourth PC, the velocity magnitude is smaller again, than those at the entrances of the first, second and third pallet columns.

Figures 6 to 9 (right plots) illustrate the static pressure distribution at different vertical sections (Section 1 to 4) of the kiln. Figure 10 (right plot) illustrate the evolution of static pressure through middle of flow at the CLHS. In the same figures there is a comparison between the k-ε and the rng k-ε turbulence models for both modes of operation.
Figure 6. Lateral velocities (left) and pressure (right) distributions in section 1.

Figure 7. Lateral velocities (left) and pressure (right) distributions in section 2.

Figure 8. Lateral velocities (left) and pressure (right) distributions in section 3.

Figure 9. Lateral velocities (left) and pressure (right) distributions in section 4.
Figure 10. Longitudinal velocities (left) and pressure (right) distributions along the centerline of the drying kiln.

Figure 11. Comparison of $\delta$ between $k$-$\varepsilon$ and rng $k$-$\varepsilon$ turbulence model in CLHS.

Figure 12. Comparison of Turbulent Intensity in Section 3 and 4.

The distribution of the static pressure in the chamber, as shown at Figures 6 and 7, reflects the presence of a low velocities regime especially from the seventh row pallet and upwards. With regard to the efficiency of air ventilation and its uniformity within the whole enclosure and throughout the load, this type of airflow is undesirable, particularly in the upper space of the chamber where high levels of temperature and contaminants could be expected due to pore mixing with the inlet jet. At a distance $x/L=0.85$ approximately from the origin point of the kiln there is a relative pressure drop from $17 \times 10^3$ to $12 \times 10^3$. This drop of the static pressure is due to the presence of the screen at this location.

In Figure 11, $\delta$ represents the relative difference of longitudinal velocities calculated with rng $k$-$\varepsilon$ and $k$-$\varepsilon$ turbulence model with respect to the “rng $k$-$\varepsilon$” velocity at centerline of the kiln for loaded and unloaded modes of operation. When the kiln operates “unloaded” the velocity predictions in terms of the parameter $\delta$ reaches 55% and in contrary when the kiln operates in “loaded” mode, the velocity predictions are independent of the turbulence model.

At the end, Figure 12 illustrates the turbulent intensity in vertical Sections 3 and 4. At the unloaded configuration the turbulent intensity is 64% in section 4, in reverse of the loaded arrangement where the turbulent intensity reaches almost 17%. This difference between the two configurations could be explained by the presence of the pallets.

4 CONCLUSIONS

Air flow patterns in an industrial wood pallet drying kiln have been investigated using a cfd in a loaded and unloaded mode of operation. In order to solve the computational difficulties for simulation of an industrial kiln, a simplified procedure has been developed in which the pallet was replaced by a rectangular solid. Turbulence is simulated with standard $k$-$\varepsilon$ and rng $k$-$\varepsilon$ model. The results obtained show the presence of low velocities regimes inside the pallets columns and above of them and the presence of recirculation regions in different zones of the drying kiln with the two turbulence models. Further work will focus on validating the cfd results and improve the geometry of the drying kiln with planned experiments in the near future.
ACKNOWLEDGMENTS

The authors gratefully acknowledge Mr. George Varsos from the Vencon Varsos S.A. (Athens, Greece) for his kind support and encouraging.

REFERENCES