International Journal of Mechanical Sciences 134 (2017) 28–40

Contents lists available at ScienceDirect

International Journal of Mechanical Sciences

journal homepage: www.elsevier.com/locate/ijmecsci

Effect of fluid channels on flow uniformity in complex geometry similar to lattice brick setting in tunnel kilns

Jaber H. Almutairi, Mosab A. Alrahmani, Issa F. Almesri, Hosny Z. Abou-Ziyan*

Mech. Power Eng. Dept., College of Technological Studies, PAAET, Kuwait

ARTICLE INFO

Keywords:
Tunnel kilns
Fluid flow
Flow malfunction
Flow separation
Flow uniformity
Flow distribution around an array of rectangular bodies

ABSTRACT

This paper reports the results of air flow in tunnel kilns using a 3D computational fluid dynamics (CFD) model. A mesh sensitivity analysis was performed, and the model was validated against experimental results. Three turbulence models that are available in general purpose commercial CFD tools, namely k–ω, standard k–ε, and RNG k–ε were tested, and the k–ω model provided results that were the closest to the experimental data. The numerical results demonstrated fluid flow malfunctions in tunnel kilns, using the size of fluid channels as for the experiment, including an intriguing phenomenon of long flow separation zone behind the brick setting. Homogenous flow was achieved by optimizing the dimensions of the side wall channels, the column channels, the ceiling channel, and the extension channels. The uniform flow, in tunnel kilns, is accomplished when the dimension of the column, wall, ceiling and extension channels was 0.5, 0.25, 0.2 and 0.36 of the column width of the brick setting. Although the pressure drop in the flow uniformity case is larger than that in the base case, the advantages of short baking time and enhanced quantity and quality of produced bricks far exceed the increase in pumping power.

1. Introduction

Bricks are a commercial ceramic product made of clay minerals. During the manufacturing process, the ceramic material goes through the steps of shaping (molding), drying, and firing [1–3]. Tunnel dryers and tunnel kilns usually are used to dry and bake the bricks, and the rates of heat and mass transfer control the speed of the process and the quality of the products. A tunnel kiln consists of three main zones, i.e., the preheating zone, the firing zone, and the cooling zone. Green (untreated) bricks enter the preheating zone of the kiln, and they are heated by the flue gas that comes from the firing zone. Thus, the process could be considered as a countercurrent flow heat exchanger. In the firing zone, the green products are heated to the desired temperature to produce the bricks. Then, the bricks are transported continuously to the cooling zone where their temperature is reduced before they leave the kiln [4].

Tunnel kilns are tunnels that have lengths that range from 90 to 180 m, widths that range from 3 to 4.24 m, and heights that range from 2.4 to 3.6 m. Tunnel kilns have been used since the beginning of 20th century when they replaced other types of kilns, and they became the most popular type of kiln by the middle of the 20th century. The designs of kilns are still being assessed and improved to enhance their efficiency and productivity and to improve the quality of their products. Therefore, experimental, analytical, and numerical investigations of tunnel kilns and related parameters have been emerging over the years. The convection heat transfer coefficients between brick settings and an air or gas flows were determined by Dugwell and Oakley [5] and by Abou-Ziyan [6]. Dugwell and Oakley built a laboratory experiment with a scale of 1:10 of the real kiln with chrome-magnesite blocks. Abou-Ziyan used a 1:4 laboratory scale experiment for six different brick settings. Both investigations provided Nusselt number (Nu) correlations similar to those used for flow in pipes to evaluate the convective heat transfer between brick settings and the air (or gas) flow. Karaush et al. [7] constructed an experiment to study the effect of thermal radiation on the setting geometry. Gol'tsova et al. [8] conducted an experimental study in which they determined that the heat loss from the walls and roof of the kiln was about 12.1%, and they indicated that additional lining and insulation were needed to reduce the heat loss to about 5–6%. Vogt and Beckmann [9] used some work reported in the literature and some dimensionless parameters to develop generalized equations for convective heat transfer in channels and in the extensions of brick settings. They stated that the non-dimensional length of the extension (extension length/equivalent diameter) and the ratio of the area of the channels to the area of the extension control the heat transfer in the extension itself.

Almeida et al. [10] presented a mathematical model and numerical solution for drying hollow ceramic bricks, and the numerical solution and the experimental results of the moisture content and temperature

---

Abbreviations: CFD, Computational Fluid Dynamics; DNS, Direct Numerical Simulation; LES, Large Eddy Simulation; MC, Mega Cells; RANS, Reynolds Averaged Navier–Stokes.

* Corresponding author.

E-mail addresses: hz.abouziyan@paaet.edu.kw, hosnyaz@hotmail.co (H.Z. Abou-Ziyan).

https://doi.org/10.1016/j.ijmecsci.2017.10.001
Received 17 April 2017; Received in revised form 5 September 2017; Accepted 3 October 2017

Nomenclature

cp	specific heat of air (J/kgK)
k	thermal conductivity (W/mK)
Nu	Nusselt number
p	pressure (Pa)
Re	Reynolds number based on the free-stream conditions and hydraulic diameter
S_T	source term
t	time (s)
T	temperature (K)
u	x-velocity (m/s)
v	y-velocity (m/s)
w	z-velocity (m/s)
x	x-coordinate
y	y-coordinate
z	z-coordinate
Δp	pressure drop (Pa)
μ	dynamic viscosity (Pa.s)
ρ	fluid density (kg/m^3)

were in good agreement. The authors reported better drying for the first row than the last row because the air has a low absolute humidity at the first row, but it becomes partially saturated as it penetrates the subsequent rows of the product.

Tahirbegović et al. [11] presented a mathematical model for the calculation of resistance to heat transfer at the cross-flow of gas in tunnel ovens used for the production of ceramic elements to be used in construction. Their results showed that the resistance to heat transfer at the cross-flow of gas was determined by several factors, including the method of setting the processed elements, the setting height, the available cross passages, densities of the elements, heating/cooling rate, and the characteristic temperature in the voids between set elements. However, the authors only considered the flow in 1D and they ignored the resistances in the other dimensions.

Oba et al. [4] reported a literature review that revealed a lack of studies on more complex 3D numerical models applied to tunnel kilns with results capable of identifying critical spots inside the kiln and the load. The authors reported tri-dimensional modeling of a tunnel kiln for the production of roof tiles operating with firewood and shale oil. However, to avoid the high computational cost due to the large dimensions of the numerical domain, they did not provide numerical solutions to the Navier–Stokes equations as the flow field of flue gas inside the kiln was defined. Thus, their results were focused on the temperature and heat flux profiles inside the kiln.

Brick making is considered to be one of the industries that require the extensive use of energy [6]. Intensive efforts have been directed toward reducing the energy consumption of this industry, particularly because the actual specific energy consumption is generally more than double the theoretical consumption, which is 550 kJ/kg [6]. A uniform flow distribution around the brick setting could enhance the process and achieve consistent quality for all bricks in the settings. Therefore, achieving even flow around the bricks and setting is considered to be the first step to guarantee the successful functioning of tunnel kilns. The above literature review indicated that there were no prior investigations related to achieving uniform flow in tunnel kilns. Therefore, in the present work, the 3D numerical modeling approach was used with the aim of obtaining the optimum flow channels to solve flow malfunctions in tunnel kilns. The flow channels of the kiln need to be set as functions of brick setting parameters.

With the rapid advances in the capacity and speed of computers, the CFD technique has become a powerful tool for providing detailed information about airflows and the distribution of fluid properties in various environments. CFD is capable of providing extensive details, a higher degree of flexibility, and flow visualization, and the technique is much cheaper than experimental studies. Therefore, it can be considered as a good alternative to full-scale measurements, which are expensive, time consuming, and have some limitations [12,13].

The main objective of the present work was to facilitate the improvement of air or gas flow in tunnel kilns in order to optimize their use of energy. Thus, a 3D model of an actual tunnel kiln with the lattice brick setting, which currently is the most popular arrangement, was developed. The model considers the arrangement of longitudinal and transverse bricks and the tiny stacking channels between them in the setting. Many of the works reported in the literature dealt with brick columns as solid blocks [5], which ignore the details that influence the fluid flow in kilns. The developed 3D model was used to visualize the flow in actual kilns with the existing situation, as reported in the experimental work of Abou-Ziyan [6]. Then, a parametric study was conducted to solve the flow malfunctions in the kiln by considering the effects of various flow channels, including the wall, column, ceiling, and extension (space between rows) channels. It is to be stated that 3D modeling is essential to investigate the effects of the fluid channels on the flow uniformity because of its ability to visualize the flow in all channels. The lattice setting with the improved flow in tunnel kilns should improve the quality and quantity of production because all of the bricks will be subjected to the same flow conditions. The results of this work will be useful in preparing better designs and providing better operating conditions for tunnel kilns. Energy conservation, productivity enhancement, and quality improvement of production are useful outcomes of this work. In addition, the results of flow uniformity in such complex geometries may be useful for other applications, such as cooling of electronic component arrays.

2. Numerical modeling and simulation

This section presents the numerical modeling of the air flow through the considered arrangement of the lattice bricks placed in a wind tunnel. So, the commercial CFD code (FLUENT version 14.5) was used to calculate the air flow pattern and the distributions of air velocity across the bricks. This is a general purpose CFD program based on a finite-volume discretization method. The second-order upwind scheme was used for pressure, momentum, energy, and turbulent kinetic energy. The pressure-velocity coupling algorithm, i.e. “coupled” was used, and each simulation was assumed to converge when the residuals for the energy equation were less than 10^{-8} and those for the equations of continuity and momentum were less than 10^{-7}.

The equations solved in each computational cell were the Navier–Stokes equations, as follows:

Conservation of mass (continuity equation):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i\right) = 0 \tag{1}$$

where ρ is the density of the fluid, t is time, u_i is the velocity vector components (u, v, and w), and x_i is the Cartesian coordinate axis (x, y, and z).

Conservation of momentum equation:

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i \tag{2}$$

where p is pressure, μ is the kinematic viscosity, and ρg_i is the body force in the x, y, and z directions.

Conservation of energy equation:

$$\frac{\partial}{\partial t}(\rho T) + \frac{\partial}{\partial x_i}\left(\rho u_i T\right) = \frac{\partial}{\partial x_i}\left[\frac{k}{c_p}\frac{\partial T}{\partial x_i}\right] + S_T \tag{3}$$

where T is the temperature, k is the thermal conductivity of the material, c_p is the specific heat, and S_T is the source term.

Turbulence was accounted for by time-averaging the equations mentioned above to produce the Reynolds Averaged Navier–Stokes (RANS) equations, and a turbulence model was used to solve for the additional terms that are generated when this process is used. Section 2.3 discusses