International Journal of Thermal Sciences 135 (2019) 285–297

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Unsteady investigation of the heat ventilation in a box prototype

Badis Bakri[a,b,*], Ahmed Ketata[a], Slah Driss[a], Hani Benguesmia[b], Zied Driss[a], Fareh Hamrit[b]

[a] Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax, B.P. 1173, km 3.5 Road Soukra, 3038, Sfax, Tunisia
[b] Mechanical Engineering Department, Faculty of Technologies, University of M'sila, Algeria

ARTICLE INFO

Keywords:
CFD
Unsteady state
Finite volume
Heat ventilation
Box prototype

ABSTRACT

This paper investigates the unsteady state of the heat ventilation in a box prototype with two holes. The CFD simulations were conducted using ANSYS Fluent 17.0 software, which solves the Navier-Stokes equations in conjunction with the standard k-ω turbulence model and by a finite volume discretization method. The presented results, consisting of the distribution of the velocity fields, the temperature, the total pressure and the turbulent characteristics, are very useful to determine the time required for the heating operation and to shrink the energy consumption of the buildings. The comparison between the founded results affirms that the heating time presents a straight effect on the velocity fields. However, for the temperature, the box prototype requires more time and more energy to warm up. In our application, we confirm that the duration of 30 s is sufficient to allow the heating of the box prototype. Indeed, the numerical results compared using the experimental data developed in our laboratory confirms the validity of the numerical method. The good agreements validate the considered computational method.

1. Introduction

The energy use worldwide is increasing every year. For example, the consumption has more than doubled in the last 40 years. The housing and service sector is one of the largest energy users on the world market. To shrink the energy consumption, the use of efficient systems is very crucial [1,2]. In this context, Driss et al. [3] developed a new system occupied by a solar patio. This system improved the microclimate of the building. In other work, Driss et al. [4] proposed an outlining environment suitable building, to improve the thermal comfort. Rauf and Robert [5] studied the life cycle in a residential building for a life range of 150 years. Premrov et al. [6] considered a timber-frame house, taking into accounts the climate data for three different European cities. Zhang et al. [7] evaluated the overall performance of eight prevalent and proposed models for simulating airflows in enclosed environments. Soni and Aliabadi [8] compared steady-state inspiratory and unsteady flows with an inlet Reynolds number of 319 at an idealized ten-generation bronchial tube model via large-scale CFD simulations. Evergren et al. [9] studied both steady and unsteady flow through a three-generation system of non-symmetric bifurcations, and they confirmed that the steady-state solution is not representative of the unsteady cases. Calay et al. [10] performed a numerical study on the unsteady respiratory airflow dynamics within a human lung based on a three-dimensional asymmetric bifurcation model by computational fluid dynamics method. The numerical results for the resting and maximal exercise breathing conditions indicated that the airflow is strongly dependent on the geometry and Reynolds number. Orhan et al. [11] solved the unsteady Navier-Stokes equations, governing the flow under Boussinesq approximation, with the vorticity-stream function formulation using the finite difference method. The development of the flow and temperature fields following these temperature changes are determined numerically. Beak et al. [12] observed some fluctuation in the overall heat transfer characteristics for a certain time. Ultimately, the radiation was found to augment the heat transfer rate, which led to reduce the time required for the flow to reach the steady state. Zhu et al. [13] studied numerically transient laminar natural convection of air in a tall cavity. The availability of the numerical algorithm was also assessed by considering the natural convection of air in a square cavity which is differentially heated from side walls. Jun et al. [14] demonstrated that the transient temperatures at the heaters may become higher than their steady-state values. The study emphasizes that the transient-stage temperatures at the heaters can exceed the corre-

* Corresponding author. Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax, B.P. 1173, km 3.5 Road Soukra, 3038, Sfax, Tunisia.
E-mail address: badisbakri@yahoo.fr (B. Bakri).

https://doi.org/10.1016/j.ijthermalsci.2018.09.023
Received 2 April 2018; Received in revised form 4 August 2018; Accepted 17 September 2018
1290-0729/

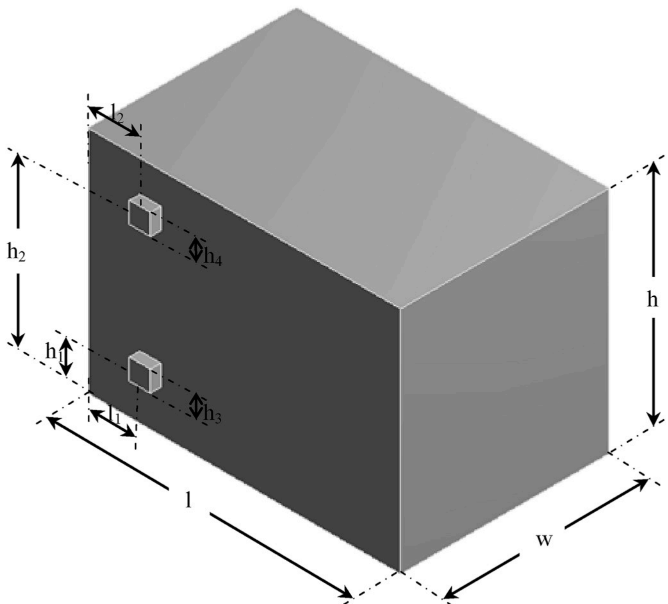

Fig. 1. Geometrical arrangement.

Fig. 2. Anemometer type AM 4204.

Table 1
Characteristics of the anemometer.

Description	Anemometer type AM 4204
Manufacturer	Lutron
Probe type	Telescopic
Measurement parameters	Air velocity, temperature, gaz flow
Resolution	Air velocity 0,1 m s^{-1}
	Temperature 0,1 °C
Precision	Air velocity 5%
	Temperature ± 0,8 °C
Measuring range	Air velocity from 0,2–20 m s^{-1}
	Temperature from −20 °C to +70 °C

Table 2
Standard k-ω turbulence model constants.

α_0	α_∞	α^*_∞	R_ω	R_k	σ_k	σ_ω
1/9	1.9	1.0	2.95	6.0	2.0	2.0

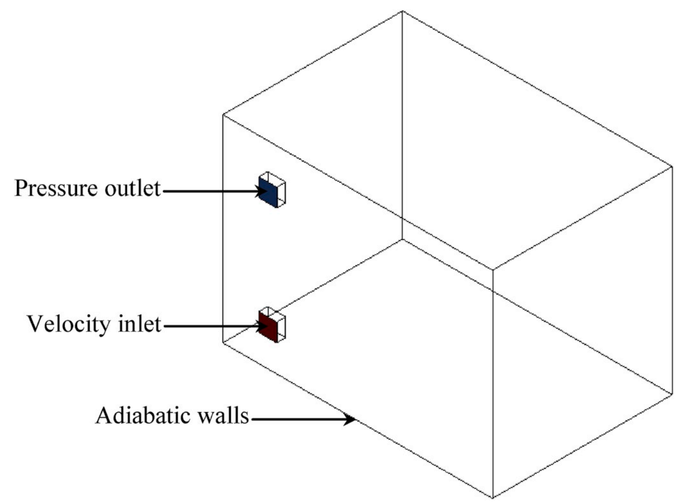

Fig. 3. Boundary conditions.

sponding steady-state values. Irfan et al. [15] study numerically a passive heating room using values of hourly averaged radiation during winter in Elazıg region in Turkey. The effect of the overall heat transfer coefficient of glass on the Nusselt number was also investigated. It was revealed that the overall heat transfer coefficient for low Rayleigh number affected the average Nusselt number more than that of high Rayleigh number. Inna et al. [16] solved numerically the unsteady conjugate natural convection problem in a square enclosure filled with a porous medium having finite thickness heat conducting solid walls at the presence of the internal heat source with constant temperature in conditions of convective–radiative heat exchange with an environment on one of the external borders. Zhang et al. [17] analyzed the back heat transfer and heat transfer resonance phenomena, and their relationships with the time-periodic flow patterns and temperature distributions. The findings are helpful to the understandings of the fluid flow and heat transfer mechanisms in the related enclosure configurations. Semen et al. [18] developed a numerical analysis of transient laminar natural convection and surface radiation in a closed cavity with heat-conducting solid walls of finite thickness and a local heat source of constant temperature in convective heat exchange with an environment. A two-dimensional numerical analysis of combined heat transfer in an air-filled square enclosure having heat-conducting solid walls of finite thickness and a local heat source in conditions of convective heat exchange. Noh-Pat et al. [19] presented a numerical study of the unsteady conjugate heat transfer for a square cavity with a semi-transparent (glass) wall. The cases with or without solar control film on the semitransparent wall were studied from 8:00 to 18:00 h; a time with the multicellular flow than the cavity without film. Armengol et al. [20] studied the effects of variable properties of air in the transient problem of the differentially heated square cavity. The investigation is therefore put forward covering additional effects under regard to the principle of energy conservation, such as the time evolution of the total energy and its components for both transient regime and steady-state cases. Mikhail