International Communications in Heat and Mass Transfer 94 (2018) 106–114

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Test of turbulence models for heat transfer within a ventilated cavity with and without an internal heat source

A. Piña-Ortiz[a], J.F. Hinojosa[a,*], J.P. Xamán[b], J.M.A. Navarro[a]

[a] Departamento de Ingeniería Química y Metalurgia, Universidad de Sonora, Hermosillo 83000, Sonora, Mexico
[b] Centro Nacional de Investigación y Desarrollo Tecnológico, CENIDET-DGEST-SEP, prol. Av. Palmira S/N. Col. Palmira, Cuernavaca, Morelos 62490, Mexico

ARTICLE INFO

Keywords:
Ventilated cavity
Internal heat source
Turbulence models

ABSTRACT

In this study, six of most frequently used turbulence models in computational fluid dynamics were compared with experimental data. The experimental cavity is a cube whose edge is 1 m long. The left vertical wall receives a constant and uniform heat flux, while the opposite wall is kept at a constant temperature. The rest of the walls are adiabatic. The heat source is a rectangular parallelepiped with square base 0.3 m on a side and height 0.61 m. The cavity represents a ventilated room in a 1:3 scale with multiple inlets and outlets of air, considering ventilation by ducts of an air-conditioning system. The experimental setup was built to obtain temperature profiles and heat transfer coefficients. Temperature profiles were obtained at six different depths and heights consisting of 14 thermocouples each. It was found that the lowest average of percentage differences for the case with the heat source turned on corresponds to realizable k-ε turbulence model (rkε) with 0.84%.

1. Introduction

At recent years, there has been a sustained increase in energy consumption at arid regions around the world also known as desert climate. In this climate, there are large diurnal temperature variations. Therefore, due to the design of existing buildings, artificial air conditioning systems are required to achieve comfort conditions, resulting in high electricity consumption. The consequence is an ecological footprint, since most of primary electricity production around the world comes from fossil fuels that are responsible of the discharge into the atmosphere of important amounts of greenhouse effect gases, mainly carbon dioxide. Therefore, the study of buildings ventilation has become an important need to reduce the energy consumption.

On the other hand, the interaction between electronic equipment or people's heat fluxes at the interior may cause different flow and temperature patterns, therefore it must be analyzed to optimize and to reduce the electricity consumption of the ventilation systems. To understand the effect of heat transfer on airflow and temperature distribution, theoretical studies may be used considering a room as a cavity, to get a better control of the studied parameters. With the mathematical model, it is possible to predict what will happen in the system, but it is necessary to establish its predictive capacity by comparing with experimental data of the thermal system.

The investigations of heat transfer in ventilated cavities are numerous [1–32], therefore, will be described only the studies with internal heat generation. Those studies may be organized as: (a) Numerical, (b) Experimental and (c) Numerical-Experimental.

(a) Numerical studies in ventilated cavities with internal heat generation.

Papanicolaou and Jaluria [18] carried out a numerical study of the mixed convection in a rectangular ventilated cavity with a discrete heat source in it. The effects of the Reynolds number, Richardson number, position of the heat source and position of the outflow on heat transfer and temperature distribution were observed for a Reynolds number from 50 to 2000. The obtained results were used to study heat removal mechanisms in practical systems. In another article, these authors [19] studied numerically a problem of mixed convection considering a cavity with two ventilation ports, conductive walls and a discrete heat source on a wall. The Reynolds number was fixed at 100 and Richardson number was varying between 0 and 10. The results show that the configuration in which the two types of convection are assisted presents higher heat transfer and lower sources temperatures.

Hsu and Wang [20] performed a numerical study about mixed convection in a ventilated cavity with discrete heat sources embedded on a vertical board which is situated on the bottom wall of the enclosure. The Reynolds number was studied from 100 to 1000. They found that when the source is located on the right surface of the board the Nusselt number is not depending neither on the variation of the

* Corresponding author.
E-mail address: fhinojosa@iq.uson.mx (J.F. Hinojosa).

https://doi.org/10.1016/j.icheatmasstransfer.2018.03.021

location of the source nor of the board. Ghasemi and Aminossadati [22] studied numerically the mixed convection in a two-dimensional ventilated cavity with discrete heat sources. They examined the effects of the number and position of the sources, the Rayleigh number from 0 to 10^7 at a Reynolds number of 100. Results show that increasing significantly Rayleigh number improves the heat transfer process in the cavity. The arrangement of sources also has a great contribution on the cooling performance but when the Rayleigh number is increased this contribution decreases.

Bilgen and Muftuoglu [23] numerically investigated a cooling strategy in a square cavity with adiabatic walls and a heat source on the left wall. The Rayleigh number was studied from 10^3 to 10^7 and the Reynolds number from 10^2 to 10^3. The authors observed that the optimal position of the source is almost insensitive to variation in Rayleigh and Reynolds numbers, but it is strongly affected by the arrangements on the ventilation ports. It was found that the highest cooling performance is given by placing the air outlet on the upper left part of the cavity. Rodriguez-Muñoz et al. [24] studied numerically the combined effect of heat generation produced by a human being and the mixed turbulent convection with thermal radiation, as well as the CO_2 production from respiration in a rectangular ventilated room. The results showed that the ventilation reduces the average temperatures in the room between 4 °C and 5.5 °C, while the thermal radiation increases the average temperature between 0.2 °C and 0.4 °C.

Biswas et al. [25] numerically studied thermal management in a ventilated enclosure undergoing mixed convection by dividing the entire heating element into multiple equal segments and by positioning them appropriately on vertical side walls, namely at bottom, middle or top positions. Analysis of segmental heating and whole heating are conducted for different Richardson number (0.01–100) and Reynolds number (50–200). Nine positional configurations of bi-segmental heating reveal the possibility of significant enhancement in heat transfer.

(b) Experimental studies in ventilated cavities with internal heat generation.

Ajmera and Mahur [26] performed an experimental investigation of mixed convection in multiple ventilated enclosures with discrete heat sources. The flow velocity and applied heat flux were varied. The experimental investigations were executed for a range of Reynolds number and Grashof number of $270 \le Re \le 6274$ and $7.2 \times 10^6 \le Gr \le 5.5 \times 10^7$, whereas the Richardson number lied in the range of 0.201–571. Different correlations were proposed for Nusselt number within the range of considered parameters.

(c) Experimental and numerical studies in ventilated cavities with internal heat generation.

Radhakrishnan et al. [21] reported a numerical and experimental work about turbulent mixed convection in a ventilated cavity with adiabatic walls and a discrete heat source inside. The k–ε (RNG) model is adopted for the turbulence closure in the two-dimensional numerical study. Correlations were developed for the average Nusselt number and the maximum dimensionless temperature occurring in the heat source, in these parameter ranges: $1200 \le Re \le 10{,}000$ and $0.003 \le \mathrm{Ri} \le 0.2$. The authors concluded that a combined experimental and numerical investigation would significantly reduce the effort required to optimize the cooling of electronic equipment.

The literature review indicates very scarce numerical and experimental studies analyzing the effect of an internal heat source on turbulent mixed convection in a ventilated cavity. Nevertheless, the study of turbulent heat transfer in a ventilated cavity with an internal heat source is relevant for an adequate thermal design of a building. Considering the above, this paper presents an experimental and numerical study of three-dimensional turbulent mixed convection in a ventilated cavity without and with an internal heat source (turned on and turned off). The cavity has multiple inlets and outlets of air, considering ventilation by ducts of an air-conditioning system. In addition, the cavity receives a heat flux in one vertical wall (representing incoming heat from the exterior). The experimental setup was built to obtain temperature profiles and heat transfer coefficients. The numerical results obtained with six turbulence models, are compared with experimental results and percentage differences are computed.

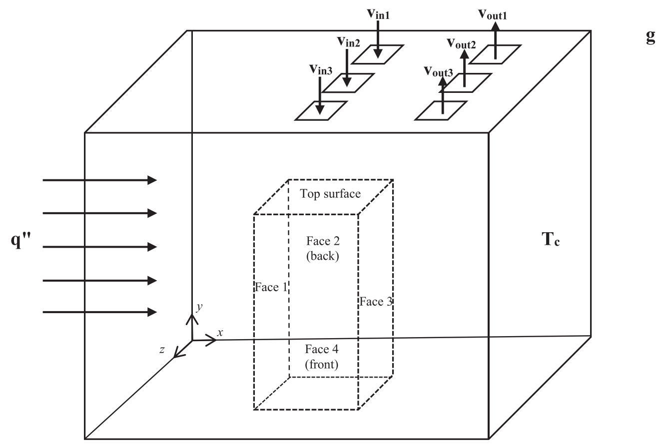

Fig. 1. Physical model of the studied cavity.

2. Physical and mathematical model

2.1. Physical model

The study of turbulent mixed convection was performed in the cubic cavity shown in Fig. 1. The cavity represents a ventilated room in a 1:3 scale. The dimensions of the system are as follows: $L_x = L_y = L_z = 1.0$ m and it consists of one vertical wall ($x = 0$) receiving a constant and uniform heat flux, while the vertical facing wall ($x = L_x$) was kept at a constant temperature T_c (298 K). The remaining walls were assumed as adiabatic. Every wall of the cavity was covered with a film of polished aluminum to minimize the thermal radiation exchange. The thermal fluid is air. The dimensions of the inlet and outlet are $l_x = l_y = 0.08$ m and their positions are described in Table 1. The air at constant temperature enters the cavity with velocities of 0.5 or 1.3 m/s. At the center of the cavity, there is a parallelepiped with a height of 0.61 m and a depth and length of 0.30 m with electrical heaters in every surface to represent the generated heat by a person in a room. The fluid flow was assumed as turbulent and because the temperature differences are < 30 K, the Boussinesq approach was considered valid.

2.2. Mathematical model

The considerations for the mathematical model are: steady state, Newtonian fluid, turbulent flow regime, negligible viscosity dissipation and the use of the Boussinesq approximation. The time averaged

Table 1
Inlet and outlet positions.

	Position (x, y, z)		
Inlet 1	$0.46\ \mathrm{m} \le x \le 0.54\ \mathrm{m}$	$y = 1.0$ m	$0.21\ \mathrm{m} \le z \le 0.29\ \mathrm{m}$
Inlet 2	$0.46\ \mathrm{m} \le x \le 0.54\ \mathrm{m}$	$y = 1.0$ m	$0.46\ \mathrm{m} \le z \le 0.54\ \mathrm{m}$
Inlet 3	$0.46\ \mathrm{m} \le x \le 0.54\ \mathrm{m}$	$y = 1.0$ m	$0.71\ \mathrm{m} \le z \le 0.79\ \mathrm{m}$
Outlet 1	$0.71\ \mathrm{m} \le x \le 0.79\ \mathrm{m}$	$y = 1.0$ m	$0.21\ \mathrm{m} \le z \le 0.29\ \mathrm{m}$
Outlet 2	$0.71\ \mathrm{m} \le x \le 0.79\ \mathrm{m}$	$y = 1.0$ m	$0.46\ \mathrm{m} \le z \le 0.54\ \mathrm{m}$
Outlet 3	$0.71\ \mathrm{m} \le x \le 0.79\ \mathrm{m}$	$y = 1.0$ m	$0.71\ \mathrm{m} \le z \le 0.79\ \mathrm{m}$