Applied Thermal Engineering 94 (2016) 159–169

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Combined convective heat and airborne pollutant removals in a slot vented enclosure under different flow schemes: Parametric investigations and non unique flow solutions

Xiu-Hong Ren [a], Jiang-Tao Hu [a], Di Liu [b], Fu-Yun Zhao [a,*], Xiao-Hong Li [a], Han-Qing Wang [c]

[a] School of Power and Mechanical Engineering, Wuhan University, Wuhan, 430072 Hubei, China
[b] Department of Building Service Engineering, The Hong Kong Polytechnic University, Hong Hum, Kowloon, Hong Kong
[c] School of Civil Engineering, Hunan University of Technology, Zhouzhu, 412007 Hunan, China

HIGHLIGHTS

- Combined convective heat and airborne transports under different flow schemes.
- Natural and forced convection dominated regimes were identified with transition.
- Dual solution branches were sustained for the transitional mixing flow scheme.
- Rest solutions evolving from motionless flows coincided with other solution branch.
- Heat and species lines were presented to delineate heat and mass transport structures.

ARTICLE INFO

Article history:
Received 10 July 2015
Accepted 22 October 2015
Available online 2 November 2015

Keywords:
Slot ventilated enclosure
Combined forced and natural convections
Convective flow schemes
Numerical simulations

ABSTRACT

This paper reports a numerical study of mixed convection on a heated and polluted strip within a slot ventilated enclosure in which the displacement and mixing flow schemes are considered. Contours of streamfunction, heatfunction, and massfunction are presented to clearly scrutinize the mechanism of heat and airborne pollutant transports. For the displacement flow scheme, thermal Nusselt and pollutant Sherwood numbers under different Reynolds numbers remain almost constant as the value of Gr/Re^2 decreases down to the regime of forced convection dominated. However, as *Ar* increases up to the regime of natural convection dominated, both *Nu* and *Sh* increase sharply with *Ar* (Gr/Re^2). Similar trends could be observed for the situation of mixing ventilated flow scheme. In the mixing scheme, non unique steady flow solutions could be observed for the range of transitional flow regime. Upward solutions, downward solutions and rest solutions have been exemplified with varying Gr/Re^2. Dual solution branches could be sustained at the range of $39.0 \le Gr/Re^2 \le 6.0 \times 10^3$, while the rest solutions obtained from rest states were completely coinciding with former continuous solutions. The present work could be significant for the natural optimization and passive control of heat and pollutant removals from the electronic boxes or building enclosures.

1. Introduction

Mixed convection is that type of heat transfer in which there is an intrinsic interaction between natural convection and forced convection. Mixed convective heat transfer in open cavities occurs in many technological and industrial processes, such as cooling of electronic components, nuclear reactors, solar receivers, chemical processing, thermal removal and pollution control in buildings.

In the past decades, mixed convection heat transfer has been a great interest due to its wide applications in practical engineering. Papanicolaou and Jaluria [1,2] numerically studied two-dimensional laminar mixed convection in a rectangular enclosure with a discrete heat source mounted on the wall. Hsu et al. [3] and How and Hsu [4] have investigated mixed convection in a partially divided rectangular enclosure. Yilbas et al. [5] discussed the effect of an internal volumetric heat-generating and conducting solid body on the mixed convection in a square cavity. Angirasa [6] and Raji and Hasnaoui [7] have numerically studied laminar mixed convection in a two-dimensional enclosure heated from one sidewall and submitted to an either aiding or opposing jet. Manca et al. [8] presented a numerical analysis of laminar mixed convection in an open cavity with a heated wall bounded by a horizontally insulated plate, where the authors considered three heating modes: assisting flow, opposing flow and heating from below. Manca et al. [9] tested experimentally a similar problem for the case of assisting forced flow configuration. Singh and Sharif [10] have investigated mixed convection cooling of

* Corresponding author. Tel.: 0086 027 6877 6651.
E-mail addresses: fyzhao@whu.edu.cn, zfycfdnet@163.com (F.-Y. Zhao).

http://dx.doi.org/10.1016/j.applthermaleng.2015.10.093
1359-4311/

a two-dimensional rectangular cavity with differentially heated side walls. They observed that maximum cooling effectiveness is achieved if the inlet is kept near the bottom of the cold wall while the outlet is placed near the top of the hot wall. Bhoite et al. [11] studied numerically the mixed convection flow and heat transfer in a shallow enclosure with a series of block-like heat generating components, regarding the broad range of Reynolds number, Grashof number and thermal conductivity ratio of solid to fluid. Tripathi and Moulic [12] investigated air circulation and temperature distribution in an air conditioning room, concerning particular locations of inlet and outlet ports on the opposite walls. Effects of *Re* and *Gr* numbers on the fluid flow and temperature fields were analyzed. Tmartnhad et al. [13] numerically studied mixed convection in a trapezoidal cavity with two changeable openings. Their results showed that the flow structure and heat transfer rate depend significantly on the inlet opening site. Bilgen and Muftuoglu [14] investigated numerically cooling strategy by mixed convection in a square cavity having two ventilation ports and with a discrete heater at its optimum position. They found that the optimum heater position could be determined by maximizing the global conductance at different Rayleigh and Reynolds numbers. Moraga et al. [15] analyzed numerically mixed convection in a square cavity inner-centered by a rectangular container, with parameters of $Gr/Re^2 = 1$, $Re = 200$ and 500. Natural convection heat transfer rate inside the inner water container being cooled by the surrounding air mixed convection is higher than that inside the inner air container being cooled by the surrounding water. Stiriba et al. [16] numerically analyzed a mixed convection flow over a three-dimensional cavity, and they investigated the complex interaction between the induced stream flow at ambient temperature and the buoyancy-induced flow from the heated wall over a wide range of the Grashof number (10^3–10^6) and two Reynolds numbers $Re = 100$ and 1000. Their results indicated that the flow becomes stable at moderate Grashof number and exhibits three-dimensional structure, while the flow becomes unsteady for both high Reynolds and Grashof numbers due to the mixed convection that come into effects. Arce et al. [17] presented a steady state numerical study of combined laminar mixed convection and thermal conduction in a ventilated square room with conductive walls. Thermal Rayleigh numbers were in the interval of $10^4 < Ra < 10^6$, while Reynolds numbers were in the interval of $100 < Re < 700$. Optimum outlet position has been determined depending on the temperature distribution effectiveness index (overall ventilation effectiveness) and the heat removal from the enclosure. Rahman et al. [18] have investigated transient mixed convection in a ventilated cavity with a partially heating and humidifying source. Three different configurations were tested according to the location of outlet port. Their results indicated that the heat and mass transfer rates could be maximized when the outlet port located near the top of the left vertical wall for $Gr = 10^7$, whereas they showed almost invariant for low *Gr* values. Fontana et al. [19] have reported three dimensional characteristics of mixed convection in partially open cavity with internal heat sources. Different values for thermal Rayleigh numbers respectively associated with internal heat source intensity and with temperature difference between the hot and cold walls were evaluated in order to define the set of optimum flow structures and energy distributions. Roy et al. [20] analyzed mixed convection in a square enclosure involving isothermally hot bottom wall, cold side walls and adiabatic top wall, which compared the fluid flow and heat distribution effect for various moving walls. The simulation results indicated that the fluid flow and heat flow distributions inside the enclosure were strongly influenced by the direction and strength of the motion of wall(s).

From the aforementioned literature review, unique flow structures were modeled in the regime of mixed convection, while multiple steady flows of mixed convection in the slot vented enclosure have not been addressed yet. Actually, multiple steady state flows have been already observed in the civil and industrial processes. Hunt and Linden [21] investigated the airflow in a naturally ventilated building driven by a combination of both wind and buoyancy (stack) forces. Their researches have shown that an opposing wind leads remarkably to the possibility of three steady flow results with same boundary conditions. Similar flow phenomena have been analytically investigated by Gladstone and Woods [22], Hunt and Linden [23], Chenvidyakarn and Woods [24], and Coomaraswamy and Caulfield [25], where multiple steady state flows were observed in the natural ventilated building enclosures driven by thermal buoyancy force, or combined wind and thermal buoyancy force. Zhao et al. [26] and Liu et al. [27] investigated the multiple steady state flows in the slot vented enclosure induced by the forced convective flows, respectively by the numerical methods and reduced scale model experiments.

The present study is dedicated to the numerical investigations of the non unique steady flow states in an enclosure with two mechanically vented ports and one free vented port, where one concentrated heat and pollutant strip is centered in the enclosure floor. This model could represent the process of heat and pollutant removal from an electronic chip, heat and moisture removal from an industrial hall, to name just a few. In the following sections, two representative flow types, i.e. displacement and mixing flow schemes, will be respectively presented and discussed. Fluid, heat and species transport structures are presented respectively by the streamlines, heatlines, and masslines. Broad range of enclosure ventilated Reynolds number and thermal Grashof number will be studied regarding the different ventilated flow forms.

2. Model description and mathematical formulations

The physical model and coordinate system under consideration are schematically shown in Fig. 1. It is a rectangular partial enclosure of longitudinal width W and altitude height H. The three-dimensional enclosure is assumed to be large so that a two-dimensional enclosure analysis could be applied. The heat and pollutant source of length S_{hps} is positioned centrally on the floor, maintaining constant heating and polluting fluxes q'' and j''. Two bottom ports are respectively located on the left and right sides, being of same size, S_{low}. One top port, size of S_{upp}, is positioned centrally on the ceiling. Under the scheme of displacement fluid flow, as illustrated in Fig. 1a, mechanical fans could be assumed installing on the bottom ports and supply outdoor air of velocity u_{in}, temperature t_{in} and pollutant concentration c_{in}, whereas ceiling port is the exhaust one. As the mixing fluid flow mode is operated, as shown in Fig. 1b, ceiling port is delivering ambient fresh air with u_{in}, t_{in} and c_{in}, while the bottom ports become the outlet. The enclosure sides, excluding the opening ports and the heat and pollutant source, are assumed to be impermeable and perfect thermal insulation.

Radiation heat transfer and viscous heat dissipation are not included herein. The flow field is assumed to be incompressible with constant properties. The combined air, heat and species transport model considered here assumes steady and laminar flow. The working fluid mixture is of constant properties, being independent of the air temperature, except the fluid mixture density variations in the vertical momentum source term, for which the Boussinesq approximation is applied [28]. Vertical temperature gradients are imposed on the fluid, and the flow is initiated and evolves under the action of forced flow and the thermal buoyancy force. The mass buoyancy due to species concentration gradients is not considered here.

2.1. Governing differential equations

Concerning the ranges of governing parameters in the present work, H, u_{in}, Δt ($q''H/\lambda$), and Δc ($j''H/D$) would be adopted as characteristic scales for length, velocity, temperature and species concentration, respectively. Depending on the aforementioned assumption and characteristic scales, the equations that describe the conserva-