International Journal of Heat and Mass Transfer 117 (2018) 388–411

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Conjugate heat and species transport in an air filled ventilated enclosure with a thermo-contaminated block

Neha Gupta, A.K. Nayak *, Sumit Malik

Department of Mathematics, Indian Institute of Technology Roorkee, Roorkee 247667, India

ARTICLE INFO

Article history:
Received 10 May 2017
Received in revised form 4 October 2017
Accepted 6 October 2017

Keywords:
Ventilated enclosure
Thermo-contaminated block
Cooling efficiency
Average fluid temperature
Thermal & contaminant effects

ABSTRACT

The flow visualization, thermal and solutal effects in a ventilated enclosure around a thermo-contaminated square block is numerically modeled. Simulations are performed for different locations and different sizes of the thermo-contaminated block with inlet and outlet port along the vertical walls where coolant air is supplied and contaminated air is flush out. The block is maintained with higher temperature and species concentration compared to the injected cold fluid. The walls are assumed as impermeable and adiabatic to heat and solute. The heat and species transfer rate along the surface of the block is compared for different values of Richardson number, Reynolds number, buoyancy ratio and block positions. Cooling efficiency inside the enclosure and average fluid temperature is calculated for different physical flow parameters to find the most suitable size and position of the block in order to obtain the maximum heat and mass transfer rate inside the enclosure. It is found that maximum cooling inside the enclosure is obtained when thermo-contaminated block is placed near the outlet port.

1. Introduction

The combined effect of heat and species transfer in ventilated enclosure is receiving significant attention because of its great interest in the phenomena of many practical engineering problems such as solar collectors, heat exchangers, space heating, chemical reactors, energy storage, equipment cooling and biomedical engineering [1,2]. Many heat sources within a building (people, electronic equipment, heat or pollutant generating obstacles) can be considered as localized and understanding the way in which they stratify a space is very essential to design the efficient ventilation schemes. There are many types of airborne contaminants in buildings and are broadly classified into two types: gaseous and particulate [3]. In the present paper we are trying to consider the gaseous kind, which are usually considered as passive contaminants and followed exactly the air currents in space. Some of these common gaseous contaminants found in building ventilation are carbon monoxide, carbon dioxide, nitrogen oxide, ozone moisture, sulphur dioxide and its progeny. Many of these gas contaminants are combustion by-products given the proliferation of transportation and industrial sources, which is increasing concern about the levels of these contaminants in outdoor and consecutively the indoor air.

In many situations initial pollutant emission and contaminant source information is not known to us. Referring to the up-to-date safety problems of indoor air environment, the 'spying covered' pollutant sources effusing colorless and non-fragrance pollutants will pose a great danger to the occupants, whom could not easily identify those pollutant sources due to their productions or emissions are essentially of no interference, no colour and no smell. As a direct solution scheme, to detect any chemical or biological agents or infectious diseases, pollutant sensors can be installed. Therefore we should identify these fatal contaminant or heat sources merely depending on the limited information from the sparsely distributed sources, since detection of these agents are relatively expensive and bulky [4]. In this prospective, we have added discrete heat sources centrally positioned in the vertical walls with the external aided coolant buoyant flows, where displacement ventilation mode could be established. In addition thermo-contaminated blocks are positioned with different locations in the ventilated room to identify the best possible location to detect or pass the energy flux in a quickest manner. It is also assumed that the third dimension of the enclosure is large enough such that room air and heat transfer are two-dimensional. Numerous experimental and numerical works are conducted in a ventilated enclosure based on the convective effects due to the presence of obstacles or body with various isothermal conditions [9]. During the last decade, numerical investigation on double diffusive natural convection in enclosures due to differentially heated,

* Corresponding author.
E-mail address: ameeyakumar@gmail.com (A.K. Nayak).

https://doi.org/10.1016/j.ijheatmasstransfer.2017.10.028
0017-9310/

Nomenclature

A	aspect ratio
Br	buoyancy ratio
c^*	species concentration, (Kg/m^3)
c_h	concentration around the block, (Kg/m^3)
c_i	concentration at the inlet, (Kg/m^3)
c_o	reference concentration, (Kg/m^3)
c	dimensionless species concentration
D	mass diffusivity, (m^2/s)
g	gravitational acceleration (m/s^2)
Gr	Grashof Number
L	length of sides of the ventilated enclosure, (m)
H	length of sides of the thermo-contaminated block, (m)
L_x	dimensionless distance between y-axis and the center of block
L_y	dimensionless distance between x-axis and the center of block
p	Pressure (Nm^{-2})
P	dimensionless Pressure
Nu_{avg}	average Nusselt number
Pr	Prandtl number
Re	Reynolds number
Ri	Richardson number
Sc	Schmidt number
Sh_{avg}	average Sherwood number
t	time (sec)
T	temperature (K)
T_h	temperature around the thermo-contaminated block, (K)
T_i	temperature of the inlet fluid, (K)
T_o	reference temperature of the fluid, (K)
u_i	velocity of the inlet fluid, (m/s)
u^*, v^*	components of velocity in x^* and y^* directions (m/s)
u, v	dimensionless velocity components in x and y directions
$\vec{v}$	dimensional velocity vector (u^*, v^*)
$\vec{V}$	dimensionless velocity vector (u,v)
x^*, y^*	cartesian coordinates (m)
x, y	dimensionless Cartesian coordinates

Greek letters

α	thermal diffusivity ($k/(\rho C_p)$) (m^2/s)
β_T	thermal expansion coefficient (K^{-1})
β_c	solutal expansion coefficient (m^3/kg)
θ	dimensionless temperature
μ	dynamic viscosity (Pa s)
ν	kinematic viscosity, (m^2/s)
ψ	stream function
ρ	density (kg/m^3)
τ	dimensionless time

cooled or different positioning of blocks are carried out for effective cooling or heating [5,6]. The literature survey shows that for cooling of the heat generating body in which large heat fluxes are to be exhausted, is to allow a low forced convective cooling and buoyancy effect simultaneously. Fans are mostly placed at the exit or at the entrance based on the performance of the system to obtain a pleasant medium. The analysis of the above problem finally motivates to study the mixed convective heat and mass transfer in a vented cavity, however, in the present day scenario, analysis capability plays a vital prerequisite for design even if many commercial softwares are available to analyze the systems.

A mixed convective numerical study in an enclosure with a finite-size heat conducting body is carried by Hsu and How [7]. They observed that heat transfer rate is affected by the change of locations of the outflow openings and the average heat transfer rate is higher when the heat conducting body is far away from the inlet port. Shuja et al. [8] studied the heat transfer effects in a ventilated enclosure where a rectangular heated solid body is placed along the center and found that maximum variation of heat transfer is observed along the upper section of the cavity when the exit port is placed at the bottom wall. Numerical and experimental studies of mixed convection due to a heat generating element (heater) in a ventilated cavity is carried by Radhakrishnan et al. [9] by varying the heat source location to obtain the optimum thermal performance. They observed that effective cooling can be achieved when heater is placed within the main flow. Rahman et al. [10] numerically investigated the steady laminar mixed-convective flow in a ventilated square enclosure. The right vertical wall is heated and a heat conducting solid circular cylinder is placed inside the center of the enclosure to analyze the flow and heat transfer effects inside the cavity by changing the diameter of the cylinder. It is found that the heat transfer rate is increasing with the increase of Richardson number at constant cylinder diameter. In continuation of that work, Rahman et al. [11] tried to find the effect of heat transfer rate by placing a heat conducting square cylinder at different positions of the ventilated enclosure, the right vertical wall is maintained with high temperature as compared to other walls and the other walls are considered to be adiabatic. The maximum heat transfer is obtained in the forced convection dominated area when the cylinder is located near the top wall along the mid-vertical plane. Mamum et al. [12] studied mixed convection heat transfer effect in a ventilated square cavity by placing a heated hollow cylinder at the center of the cavity using finite element based Galerkin method. They found that for largest value of the cylinder diameter, maximum average Nusselt number and average fluid temperature can be obtained. Moreover, fluid temperature is increasing with the increasing value of Richardson number for fixed value of cylinder diameter. Literature suggests that for the maximum cooling of heat generating body inside the enclosure, body must be located along the main stream near the top wall and highest heat transfer rate can be achieved for increase of Richardson number. A combined numerical and experimental study is performed by Radhakrishnan et al. [13] in a ventilated cavity to obtain the convective heat transfer effects due to multiple heaters in a staggered arrangement. They evaluated the operating temperature of each heater for a wide range of dimensionless parameters and found that for high values of Reynolds number, the percentage increase in Nusselt number is low for the middle heaters compared to the heaters on either side.

A numerical investigation of mixed convection from a heated square solid cylinder located near the center of a vented cavity filled with air is made by Chamkha et al. [14]. The flow and the thermal field effect due to varying outlet positions with fixed inlet position is studied for a wide range of Richardson number, Reynolds number, locations and aspect ratios of the inner square cylinder. They observed that average Nusselt number along the heated surface of the square cylinder increases with the increase in Reynolds and Richardson number. They also concluded that average fluid temperature in the enclosure appeared to be constant for a highly buoyancy-dominated convection regime. A numerical study