International Journal of Thermal Sciences 116 (2017) 103–117

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Numerical investigation of turbulent mixed convection in an open cavity: Effect of inlet and outlet openings

Lounes Koufi [a, b, c], Zohir Younsi [b, c], Yassine Cherif [a, c], Hassane Naji [a, c, *]

[a] Univ. Artois, Laboratoire de Génie Civil et géo-Environnement (LGCgE - EA 4515), F-62400 Béthune, France
[b] FUPL, Hautes Etudes d'Ingénieur, LGCgE - EA 4515, 13 Rue de Toul, F-59000 Lille, France
[c] Univ. Lille Nord de France, LGCgE - EA 4515, F-59000 Lille, France

ARTICLE INFO

Article history:
Received 19 September 2016
Received in revised form
21 December 2016
Accepted 11 February 2017

Keywords:
Numerical simulation
Convection
Mixing ventilation
Ventilation efficiency
Thermal comfort
Turbulence modeling

ABSTRACT

This paper deals with the numerical investigation of heat transfer by mixed convection inside ventilated cavities with supply and exhaust slots, and filled with air under a steady and turbulent flow regime. Four configurations, rated A, B, C and D are considered here, according to the position of the inlet and outlet air ports: A, the inlet is on the top of the left vertical wall, while the outlet is on the bottom of the right vertical wall; B, the inlet is on the bottom of the left vertical wall and the outlet at the top of the opposite wall; C, the two slots are on the same side, i.e. the inlet is at the bottom and the outlet at the top of the left vertical wall, and D, the inlet is at the top and the outlet at the bottom of the left vertical wall. The bottom of the cavity is kept at a temperature T_H and other walls are fixed at a temperature T_C, with $T_H > T_C$. The cavity is provided with two slots: an inlet slot for introducing fresh air, and an outlet slot to extract hot air. The main aim sought here is to analyze the ventilation efficiency for temperature distribution, and fix the best configuration providing the thermal comfort targeted. We also address the influence of heating on the behavior of flow and thermal comfort, while considering different Rayleigh numbers ranging from 6.4×10^8 to 3.2×10^9. Numerical studies have been yet devoted to these configurations, using RANS simulations. The *RNG k-ε* turbulence model has been adopted for the turbulence closure, and the set of governing equations was then numerically solved via the finite volume method. The *SIMPLEC* algorithm was associated to ensure the pressure-velocity coupling. In terms of results achieved, the configuration D provides a better ventilation effectiveness for temperature distribution ε_T and ensures an even temperature in the occupied zone. As for configurations A and C, they maintain an acceptable level of heat and can be used in winter period to ensure good indoor air quality, while configuration B provides an efficiency close to unity and can be used to insure indoor air quality in temperate climate zones.

1. Introduction

Today, many people spend more time indoors (homes, schools, offices, transports, stores, etc.) [1]. Thereby, the quality of indoor environment must ensure the occupants' requirements in terms of thermal comfort and indoor air quality that can adversely affect their health. In addition, the indoor environment is affected by rising energy costs. Besides, since the 1973 oil crisis, thermal insulation has been greatly increased to reduce heat loss and greenhouse emissions. To this, exchanges between the outside and inside buildings were greatly reduced. Such confinement led to discomfort of the occupants and caused damage to the building structure. To improve the quality of the environment, ventilation has proved to be one of the most promising solutions. Its principle is to renew sufficiently and permanently stale air by fresh air. For that, investigators and designers innovative offer different strategies of ventilation to guarantee the comfort for occupants. Note that a ventilation system can be used to ensure good indoor air quality and thermal comfort (heating or air conditioning). There are two mechanical ventilation modes that are widely used. These are based on the fresh air intake velocity, namely the mixing ventilation where fresh air is blown at high velocities (turbulent jet air), and the displacement ventilation where air is introduced at low velocities [2]. Convective motion can help to determine the quality of the indoor air circulation in rooms and passenger cabins (cars,

* Corresponding author. Univ. Artois, Laboratoire de Génie Civil et géo-Environnement (LGCgE - EA 4515), F-62400 Béthune, France.
E-mail address: hassane.naji@univ-artois.fr (H. Naji).

http://dx.doi.org/10.1016/j.ijthermalsci.2017.02.007
1290-0729/

Nomenclature

C_p	Specific heat ($Jkg^{-1}K^{-1}$)
g	Gravity acceleration (ms^{-2})
h	Height of the gap for the inlet or outlet air (m)
H, L	Cavity's height and width (m)
k	Turbulent kinetic energy (m^2s^{-2})
n	Normal vector (m)
Nu	Nusselt number
p	Fluid pressure (Pa)
Pr	Prandtl number, $Pr = \nu_o/\alpha_o$
Q	Heat (W)
Ra	Rayleigh number, $Ra = (\rho_o g\beta\Delta TL^3)/(\mu_o\alpha_o)$
Re	Reynolds number, $Re = (\rho_o U_{in}h_{in})/\mu_o$
T	Fluid temperature (K)
T_C	Cold temperature (K)
T_H	Hot temperature (K)
T_0	Reference temperature, $T_o = (T_H + T_c)/2 \quad (K)$
T'	Fluctuating temperature (K)
u, w	Horizontal and vertical velocities (ms^{-1})
u^*, w^*	Dimensionless velocity components, $(u^*, w^*) = (u, w)/\sqrt{g\beta\Delta TL}$
u', w'	Fluctuating velocity components (ms^{-1})
U	Velocity modulus (ms^{-1})
X, Z	Horizontal and vertical coordinates (m)
X^*, Z^*	Dimensionless coordinates, $(X^*, Z^*) = (X, Z)/L$

Greek symbols

α	Thermal diffusivity (m^2s^{-1})
τ_a	Local age of fluid (s)
β	Thermal expansion coefficient (K^{-1})
ΔT	Characteristic temperature difference, $\Delta T = (T_H - T_C)(K)$
ε	Turbulent energy dissipation (m^2s^{-3})
ε_T	Ventilation effectiveness for temperature distribution
λ	Thermal conductivity ($Wm^{-1}K^{-1}$)
μ	Dynamic viscosity ($kgm^{-1}s^{-1}$)
ν	Kinematic viscosity (m^2s^{-1})
ρ	Density (kgm^{-3})
δ	Boundary layer thickness (m)
σ	Standard deviation

Superscripts/subscripts

C	Cold
H	Hot
in	Inlet
l	Local
m	Mean
max	Maximum
min	Minimum
out	outlet
T	Thermal
t	Turbulent
O	Reference

planes, etc.). It also plays a significant role in heat exchangers and passive cooling devices for electronic equipments and computer chips. It should be noted that, in a ventilated room, natural convection (driven by buoyancy forces) and/or forced convection (due to the external forces) can take place. The combination of these modes gives rise to a mixed convection problem. To handle such a problem, numerical and experimental studies have been carried out to understand the behavior of airflow by mixed convection and the temperature distribution within ventilated rooms. In the present work, we consider a ventilated cavity often taken as a first model [3,4]. In the following, we briefly describe some works related to the subject dealt here.

Blay et al. [5] studied numerically and experimentally mixed convection inside ventilated square cavity filled with air in turbulent and steady regime. This cavity is heated from below. The authors have performed simulations for different Froude number using two low-Reynolds number k-ε models [6,7]. Their results showed the existence of a critical Froude number for which the flow inside the cavity changes direction. Chen [8] evaluated the performance of five k-ε turbulence models on the heat transfer in a ventilated cavity. The author has found that the numerical predictions of average velocities are satisfactory. In addition, he noted that fluctuations are less estimated compared to available experimental results. Also, he noticed the RNG k-ε [9] model provides more accurate results than the standard k-ε model [10]. Raji and Hasnaoui [11–13] numerically analyzed the heat transfer by mixed convection inside ventilated cavities filled with air and subjected to heat flux conditions. They evaluated the influence of radiation on heat transfer by mixed convection in a rectangular cavity with an aspect ratio of 2. They found that radiation promotes the temperature's homogenization, and reduces the maximum temperature inside the cavity. Singh and Sharif [14] conducted a numerical study of mixed convection in a 2D rectangular cavity differentially heated in laminar and steady flow regime, and equipped with inlet and outlet slots. They have shown that the position of the slots can be crucial in terms of cooling efficiency. Considering the same regime, Rahman et al. [15] studied numerically the influence of key parameters as the Prandtl (Pr), Reynolds (Re) and Richardson (Ri) numbers on the mixed convection in a ventilated square cavity. The authors found that, for low values of Ri, the rate of heat transfer is minimum along the hot wall, and for high values of Pr, the Nusselt number is enhanced. Also, the increase of Ri avoids the separation of flow while exhibiting a linear behavior. They concluded that heat transfer is strongly influenced by high values of both Pr and Ri. Radhakrishnan et al. [16] performed experimental and numerical study of mixed convection in a ventilated cavity with a heat generator. They studied the influence of the size and position of heat sources on the heat transfer performance. They found that simulations corroborate experimental results. They stated that the size, position and inclination angle of the heat source influence greatly the thermal behavior inside the room.

Xamán et al. [3] numerically investigated conjugated heat transfer in a ventilated cavity under turbulent flow regime. The authors sought to determine the optimal ventilation scenario in terms of air conditioning. They found that, for Reynolds numbers ranging from 5.10^3 to 10^4, it is the cavity for which the exit is located at the top right which ensures temperature and velocity values close to those recommended by the standard ASHRAE 55 [17]. Ezzouhri et al. [18] took over the cavity considered by Blay et al. [5] to evaluate the LES model [19] and determine the exact value of the Froude number for which two different solutions exist. Comparisons with experimental data [5] have shown that such a model provides acceptable predictions. Hinojosa and Gortari [20] conducted a numerical study of heat transfer by laminar natural convection in an isothermal open cubic cavity. The authors presented the results for different Ra numbers ($10^4 \leq Ra \leq 10^7$).