International Journal of Thermal Sciences 125 (2018) 369–380

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Transient turbulent simulation of natural convection flows induced by a room heater

Mahmoud El-Gendi

Department of Mechanical Power Engineering and Energy, Faculty of Engineering, Minia University, 61517 Minia, Egypt

ARTICLE INFO

Keywords:
Room heater
Natural convection
Compressible solver
Isothermal
Adiabatic
Transient flow

ABSTRACT

A warming room air in a cold weather is an urgent need for people to keep performing normal activities in a comfortable environment. In the current study, a heater attached to one of the room's walls induces the natural convection flow. The flow is simulated by a turbulent unsteady compressible Navier-Stokes solver at a Rayleigh number of 3.06×10^{11}. Two cases are investigated and their boundary conditions are identical except the right side where is adiabatic or isothermal. Although the adiabatic case had a significant importance in the domestic heating, it wasn't investigated in the literature. The transient results show that the hotter streams occupy the top zone where the velocity is relatively high and then move downward. As a whole, the average temperature of the adiabatic case is higher than that of the isothermal case, but the isothermal case has a higher temperature gradient.

1. Introduction

The natural or free convection flows have several domestic as well as industrial applications. The room heater is an example of the domestic applications and is used commonly because of its initial and running cost compared to alternatives. The density gradient resulting from the temperature difference induces the flow under the effect of the buoyancy or gravity force.

Several studies are carried out on the natural convection flow inside the enclosure. De Vahl Davis [1] and Barakos et al. [2] provided validation data for laminar and turbulent flow inside a square cavity at different Rayleigh numbers, respectively. Hasan et al. [3] studied the effect of frequency and corrugation amplitude of a sinusoidal enclosure on natural convection flow in the range of Rayleigh number from 10^5 to 10^8. They categorized the development of the flow into three stages: initial, oscillatory, and steady. Morsi and Das [4] calculated the flow within an enclosure with different dome configurations. The roof and square enclosures have heat transfer rate lower than those of dome-shaped enclosures. Sergent et al. [5] noticed that the discrepancy between numerical and experimental data results mainly from the improper treatment of the boundary conditions of the end walls. Nithiarasu et al. [6] noticed that Rayleigh number and width ratio changed the nature of heat transfer of simulated flow within an L-shaped enclosure. Paolucci [7] provides a concrete review about the natural convection flow inside enclosures.

Natural convection flows have several applied thermal applications. The transient natural convection flows within a square cavity induced by heated square [8] and cylinder [9] bodies were investigated. The immersed boundary method [9] treated the circular boundary of the cylinder. El-Gendi [10] attached openings to the top and bottom sides of an enclosure and found that both inlet flow angle and velocity have a dominant role in the flow pattern inside the enclosure. Bocu and Altac [11] investigated 3D natural convection flows in an enclosure by attaching pin arrays to the hot wall. Béghein et al. [12] studied numerically the particle motions in a room. They verified that the Large Eddy Simulation (LES) is a successful tool to predict the particle motions. de Dear et al. [13] wrote a detailed review about the thermal comfort research. Ma and Xu [14] investigated numerically the effect of fin position on the heat transfer along with unsteady natural convection flow for a range of Rayleigh numbers (from 10^8 to 10^{11}). Baïri et al. [15] provided a concrete review on natural convection flow and they focused on the flow inside the parallelogrammic diode cavity.

The effects of thermal radiation on the heat transfer and fluid flow in cavities with and without local heaters were investigated. From two decades, Balaji and Venkateshan studied the interaction of radiation with natural convection in a closed square cavity [16] and an open cavity [17]. Xamán et al. [18] studied three effects (conduction - radiation – convection) on laminar $10^3 \leq Ra \leq 10^6$ and turbulent $10^9 \leq Ra \leq 10^{12}$ flows. They found that neglecting radiative heat transfer results underestimations in the total heat transfer. In addition, Velusamy [19] investigated the effect of Ra and aspect ratio. Several publications [20–23] investigated the interaction effects between surface radiation and free convection in conjugate cavities heated from the bottom. The influences of several parameters such as emissivity of the

E-mail address: mahgendi@mu.edu.eg.

https://doi.org/10.1016/j.ijthermalsci.2017.12.012
Received 5 May 2017; Received in revised form 31 October 2017; Accepted 11 December 2017
Available online 21 December 2017
1290-0729/

Nomenclature

g	gravitational acceleration (ms^{-2})
H	height of the room (m)
Pr	Prandtl number, $Pr = \nu/\alpha$
Ra	Rayleigh number, $Ra = g\beta\Delta TH^3 Pr/\nu^2$
R	air gas constant
T_c	temperature of the cold surface (K)
T_h	temperature of the hot surface, heater, (K)
T_m	reference temperature, $T_m = (T_h + T_c)/2$, (K)
T	temperature (K)
u	velocity component in x-direction (ms^{-1})
u^*	dimensionless velocity component in x-direction, $u^* = u/\sqrt{g\beta\Delta TH}$
v	velocity component in y-direction (m/s)
v^*	dimensionless velocity component in y-direction, $v^* = v/\sqrt{g\beta\Delta TH}$
W	width of the room (m)
x,y	Dimensionless Cartesian coordinates
x_o,y_o	Cartesian coordinates (m)
ΔT	temperature difference, $\Delta T = T_h - T_c$

Greek symbols

α	coefficient of thermal diffusion ($m^2\ s^{-1}$)
β	coefficient of thermal expansion, $\beta = 1/T_m\ (K^{-1})$
τ	computational time
φ	converged time of isothermal case
μ	total viscosity
μ_l	laminar viscosity
μ_t	turbulent viscosity
θ	dimensionless temperature, $\theta = (T - T_c)/(T_h - T_c)$

wall surface, Rayleigh number, enclosure aspect ratio, and external heat transfer coefficient were studied.

The previous studies of natural convection flow focused on differentially heated enclosures where there are a heat source (T_h) and a heat sink (T_c). The present study investigated also the transient heating of an insulated room where there is heat source only (T_h) and this case has a significant importance in the domestic heating.

2. Studied cases

The present work investigated two cases. The first case represents a room that has a wall on the building facade and is affected by the weather temperature. We coin this case as the isothermal case based on the boundary condition along this wall ($T = T_c$). The second case represents the same room but the wall is located in the interior of the building or well insulated. We coin this case as the adiabatic case based on the boundary condition on this wall. The adiabatic case must be simulated by an unsteady solver. Fig. 1 shows the middle section of the rooms in both cases.

In both cases, the left side is adiabatic and has a heater at the bottom portion. The height of the heater is one meter. The ceiling and the floor are considered as adiabatic surfaces. The width and the height of the room are 3.6 m and 3 m, respectively. T_h represents the heater temperature and is kept constant at 353 K, and T_c represents the weather temperature (T_c = 273 k) and ΔT represents the temperature difference (ΔT = 80 K). The present temperature difference (ΔT = 80 K) violates the Boussinesq approximation (ΔT < 28.6 K) [24] so that the simulation must be carried out by a compressible solver. The fluid medium inside the room is air with a Prandtl number Pr = 0.71. Based on these conditions, the Rayleigh number is relatively high ($Ra = 3.06 \times 10^{11}$) and a turbulent solver should be used.

3. Computational modelling

3.1. Initial and boundary conditions

Fig. 1 shows the boundary conditions in the present study and Fig. 2 shows the stretched one-block H-type grid used in both cases. The grids are 122 × 122 points. The non-slip condition is implemented on all sides. The adiabatic condition is used at the top and bottom sides. The temperature of the lower portion (1 m) of the left side is kept constant ($T = T_h$). The remaining of the left side (2 m) is adiabatic. The boundary conditions for the right side are isothermal ($T = T_c$) or adiabatic for the isothermal and adiabatic cases, respectively. Initially, the air is stagnant and its temperature equals the ambient temperature ($T = T_c$). We assumed that the investigated problem is two-dimensional and the equation of state can be applied. In the isothermal case of the

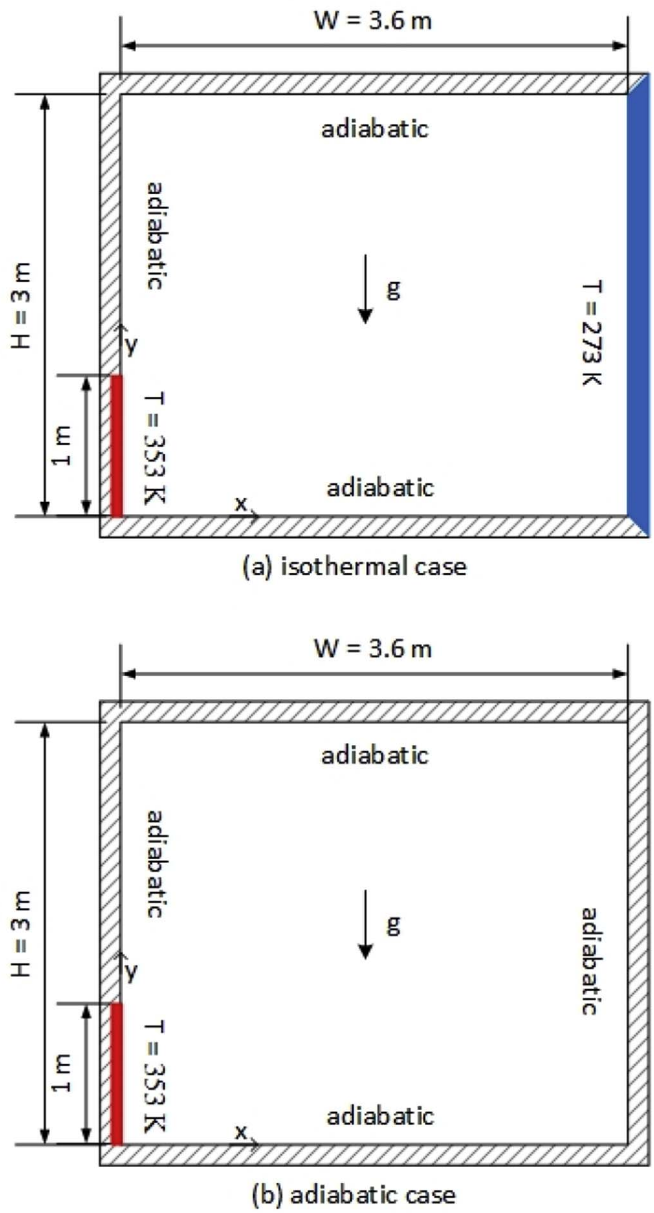

Fig. 1. Computational domain and boundary conditions for both cases.