Applied Mathematical Modelling 40 (2016) 927–941

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Non-Boussinesq approach for turbulent buoyant flows in enclosure with horizontal vent and forced inlet port

R. Harish, K. Venkatasubbaiah*

Department of Mechanical and Aerospace Engineering, Indian Institute of Technology Hyderabad, Hyderabad 502205, India

ARTICLE INFO

Article history:
Received 13 October 2014
Revised 7 April 2015
Accepted 28 May 2015
Available online 11 July 2015

Keywords:
Non-Boussinesq approach
Horizontal vent
Grashof number
Forced inlet port

ABSTRACT

The effects of forced ambient velocity on thermal plume behavior in a ceiling vented square enclosure are numerically investigated. Turbulence is modeled by unsteady Favre-averaged Navier–Stokes (UFANS) equation with Lam Bremhorst low Reynolds number $k-\varepsilon$ turbulence model. A non-Boussinesq variable density approach is used to model the density variations. Simplified Marker and Cell (SMAC) algorithm is used to solve the governing equations on collocated grid with high accuracy compact finite difference schemes. The pressure Poisson equation is solved by bi-conjugate gradient algorithm and time integration is performed with four stage Runge–Kutta method (Rk-4). The results are presented for Grashof number $Gr = 10^{11}$ and 10^{12} and Gay-Lussac number $Ga = 0.2$ and 2. The present model is valid when buoyancy effects are significant in comparison with forced convection effects. The heat transfer characteristics are analyzed by varying forced inlet velocity, inlet port size and inlet port location. The assisting flow enhances plume discharge rate and increases convective heat loss from cavity. The opposing flow weakens thermal buoyancy and minimizes convective heat loss from cavity. The present mathematical model and numerical method are in good agreement with the existing results available in the literature.

1. Introduction

The plume structures arising from the effects of thermal buoyancy and discharge rate through openings are of great importance in cooling of electronic devices, solar cavity receivers, building natural ventilation and enclosure fires. In buoyant flows, it is customary to adopt Boussinesq approximation to model thermal buoyancy force. However, for applications with high temperature difference such as nuclear reactor systems, heat generation stations, foundry processes and fire transport phenomena it is inappropriate to evaluate density variations by Boussinesq approximation. In naturally ventilated open cavities with internal heat source, heat and mass exchange through vertical and horizontal openings are generated because of density difference caused by temperature difference. The flow patterns through large vertical openings such as doorways and windows are well investigated in the literature. Brown and Solvason [1] studied the natural convection flow through single vertical rectangular openings in partitions. They determined neutral pressure level close to middle of opening height, and air flow profile through vertical openings develops stable velocity distribution. Prahl and Emmons [2] predicted the flow of fire gases through doorway of a burning room and proposed mathematical models to calculate exchange flow through vertical passages. Their model assumes

* Corresponding author. Tel.: +91 40 23016074; fax: +91 40 23016032.
E-mail address: kvenkat@iith.ac.in (K. Venkatasubbaiah).

http://dx.doi.org/10.1016/j.apm.2015.05.013

Nomenclature

H	Height of the enclosure
W	Width of the enclosure
D	Horizontal vent width
H_v	Height of inlet port
T_s	Heat source temperature
T_∞	Initial fluid temperature
g	Gravitational force per unit mass
μ	Dynamic viscosity
μ_t	Turbulent dynamic viscosity
ρ	Fluid density
k	Turbulence kinetic energy
ϵ	Turbulent dissipation rate
U, V	Dimensionless velocity components along x and y directions
τ	Dimensionless time
θ	Dimensionless temperature
K	Dimensionless turbulence kinetic energy
ε	Dimensionless turbulent dissipation rate
Pr	Prandtl number
Pr_t	Turbulent Prandtl number
Gr	Grashof number
Ra	Rayleigh number
Ri	Richardson number
Ga	Gay-Lussac's number

the presence of two gas layers in the enclosure; upper layer filled with hot fire gases and lower air layer made up of entrained ambient air.

Mercier and Jaluria [3] conducted experiments on fire induced flow of hot gas in enclosure with vertical openings. They injected hot gases into the enclosure from lower opening and observed wall plume temperature pattern inside the enclosure. Similar experiments [4,5] were carried out to predict buoyancy induced ventilation rate in compartment with sidewall opening. Abib and Jaluria [6] did numerical simulation to evaluate the buoyant flow arising from fire source in vented compartment. They analyzed thermal plume behavior and calculated the bidirectional flow rates through opening by restricting the computational domain within the cavity. Chow and Zou [7] numerically studied airflow rates through doorway in fire scenarios and the proposed empirical correlations were in good agreement with experimental data. In the literature, similar CFD simulations [8–11] were reported to predict the air movement through vertical openings.

The flow patterns through horizontal openings are more complex and unstable than vertical openings. One of the earliest studies on natural convection flows through horizontal openings was carried out by Epstein [12]. He experimentally studied the bidirectional flows through horizontal circular opening with brine and fresh as fluid medium. Epstein varied the vent aspect ratio and proposed empirical model to calculate mass flow rate. Tan and Jaluria [13–15] did experiments to study the mass flow rate through a horizontal vent in an enclosure due to pressure and density differences. They identified the critical pressure at which transition from bidirectional to unidirectional flow occurs across the vent. Recently, Venkatasubbaiah and Jaluria [16] analyzed the fire driven flow in square enclosure with single and multiple ceiling vents. They varied vent size and found that the critical Grashof number is 10^6, above this flow becomes chaotic inside the enclosure. Chow and Gao [17,18] investigated air flow patterns across horizontal vent and they identified that the ratio of buoyancy and inertia force is one of the key parameter affecting the bidirectional discharge across the opening. All the above studies concluded that flow patterns through horizontal vent is more complex and flow resistance is greater than in vertical opening. The transient behavior across the opening is due to the presence of denser fluid on top of lighter fluid and needs further understanding.

The buoyancy induced turbulent mixing is one of the key process for the growth and spread of thermal plume. Turbulent natural convection flow in differentially heated square cavity was studied by Markatos and Pericleous [19] using standard $k-\varepsilon$ turbulence model. They identified that the model is suitable for predicting buoyant flows. Chattopadhyay and Saha [20] used large eddy simulation turbulence model to simulate the flow characteristics of impinging jet over a moving plate. They have reported that turbulent kinetic energy and heat transfer from the plate increases with rise in surface velocity. Cook and Lomas [21] analyzed buoyancy driven flows in enclosure using standard $k-\varepsilon$ and RNG $k-\varepsilon$ models. They found RNG $k-\varepsilon$ model gave better results. Similarly, Stavrakakis and Markatos [22] analyzed the air flow patterns in one and two room enclosures containing a fire source. Abib and Jaluria [23] studied the turbulent penetrative and recirculating flow in partial enclosure with low Reynolds number model of Lam Bremhorst. Similarly, Davidson [24], Gupta et al. [25], Harish and Venkatasubbaiah [26,27] carried out numerical analysis on buoyancy induced turbulent flows using Lam Bremhorst model and found that this model has higher capability of predicting turbulent quantities reasonably well in regions near and away from the walls.