Fire Safety Journal 61 (2013) 160–174

Contents lists available at ScienceDirect

Fire Safety Journal

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

Transport phenomena of turbulent fire spread through compartment connected to vertical shaft in tall building

R. Harish, K. Venkatasubbaiah *

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

ARTICLE INFO

Article history:
Received 11 October 2012
Received in revised form
11 May 2013
Accepted 3 September 2013

Keywords:
Fire-induced flow
Partial enclosure
Ceiling vent
Grashof number

ABSTRACT

The purpose of this paper is to understand the growth and spread of fire through compartment connected to vertical shaft in tall building. The transport phenomena due to fire have been modeled as buoyancy induced turbulent flow in tall building. The governing equations comprise the Reynolds averaged Navier–Stokes (RANS) equations with the K–ε turbulence model in a stream function–vorticity formulation approach. The governing equations are solved by high accuracy compact finite difference schemes with the four stage Runge–Kutta method for time integration. Results are reported for two different Grashof numbers $Gr = 10^{10}$ and 10^{11}. The effects of vertical passage between compartment and shaft, multiple vertical vents and significance of flow through ceiling vent in tall building are presented. The thermal plume growth rate, the ambient entrainment flow rate and the bidirectional oscillatory nature at the vent openings are reported. A significant change in the flow behavior is observed by varying the size of vertical passage between compartment and shaft. The flow behavior through vertical and ceiling vents is presented. The flow patterns in the shaft show complex flow structure with multi-recirculating convective cells. The present results are matching with the numerical and experimental results available in the literature.

1. Introduction

The thermal plume and hot gas flow characteristics in tall buildings due to fire are important to understand the fire spread and to identify the fire hazards for risk assessment. The field modeling by computational fluid dynamics plays an important role in measuring the location of maximum temperature, pressure, velocity and smoke concentration thereby aiding fire safety measures such as location of smoke extraction system, location of fire and smoke detectors, smoke vent, physical barriers, fire escape routes and evacuation modeling. In the event of a fire accident in high rise multistoried buildings, a significant number of occupants could be exposed to smoke, toxic gases and flame from the fire which could lead to catastrophic damage endangering human life and property. During fire accidents the smoke and hot gases spread rapidly due to buoyancy effects caused by temperature differences arising from fires and due to entrainment across vents and openings. In tall buildings there are internal flow passages such as open elevator shafts, stairwells and ventilation openings such as doors and windows, which are the integral part of modern buildings. In such situations these elevator shafts which are vertical channels facilitate the fire transport phenomena due to stack effects and are connected to upper floors through vertical openings such as doors. The rate of inflow of oxygen from the ambient through these openings has a significant effect on the growth and spread of fire in buildings and to adjoining areas. Hence, the movement of smoke and fire spread is of primary concern encountered with vertical shafts for the fire safety engineering of high-rise buildings. The transport phenomena due to fire in high-rise buildings are usually buoyancy dominated and hence can be modeled as buoyancy-induced turbulent flow in tall enclosures.

The fire transport phenomena in enclosures were modeled as buoyancy induced flows and reported in [1–3]. Hanjalic et al. [4] numerically investigated a turbulent natural convection process in building connected to adjacent compartment by considering a 2D rectangular cavity with and without the presence of partitioned wall. Simulations were performed for Rayleigh number 10^{10}–10^{12} using an algebraic model for turbulent heat transport. The thermal plume behavior due to fire is modeled as buoyancy induced flow by Murgai and Emmons [5] and determined that the flow characteristics depend upon the size of fire and the ambient conditions. The horizontal vents are openings in ceilings, floors, stairwells, broken window at the top of atrium roof and smoke vent at the roof top. The flow patterns through horizontal vents are different when compared to their vertical counterparts such as doors and windows. Abib and Jaluria [6] have numerically investigated the fire induced flow through a vertical opening in enclosure. The fire transport phenomena are modeled as natural

* Corresponding author. Tel.: +91 4023016074; fax: +91 4023016032.
E-mail address: kvenkat@iith.ac.in (K. Venkatasubbaiah).

0379-7112/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.firesaf.2013.09.005

Nomenclature

H_1	height of the rectangular enclosure
W_1	width of the rectangular enclosure
H	height of the tall shaft
W	width of the tall shaft
T_s	heat source temperature
T_∞	initial fluid temperature
g	gravitational force per unit mass
t	time
X, Y	dimensionless x and y coordinates
u, v	velocity along x and y directions
β	coefficient of thermal expansion
α	thermal diffusivity
ν	kinematic viscosity
ν_t	turbulent eddy viscosity
Gr	Grashof number
Ra	Rayleigh number
Pr	Prandtl number
Pr_t	turbulent Prandtl number
k	turbulent kinetic energy
ε	turbulent dissipation rate
ψ	stream function
ω	vorticity
Ψ	dimensionless stream function
Ω	dimensionless vorticity
θ	dimensionless temperature
U, V	dimensionless velocity components along X and Y directions
τ	dimensionless time
K	dimensionless turbulent kinetic energy
ε	dimensionless turbulent dissipation rate

convection flows and their studies were limited to the laminar flow regime. Later Mercier and Jaluria [7] have extended a similar experimental study in open vertical shaft with height to width aspect of three. Smoke and hot gases are injected into the enclosure through a lower opening and the resulting downstream wall plume along the enclosure wall and flow fields are reported. Earlier studies were limited to flow through vertical openings in the enclosure.

Atkinson [8] performed experiments to understand the dynamics of smoke movement in an enclosure with horizontal vent and observed rapidly rotating smoke rolls near the ceiling. Later experimental studies on the mass flow rate through horizontal vent in an enclosure due to pressure and density differences were reported by Tan and Jaluria [9]. They found that in the absence of pressure difference but with heavier fluid overlying lighter fluid, a bidirectional flow arises across the vent due to buoyancy effects. Recently Venkatasubbaiah and Jaluria [10] performed the computational study on enclosure fire with single and multiple horizontal vents within the laminar flow regime and found that the critical Grashof number is 10^6, above this the flow becomes chaotic nature in the enclosure.

Although a large amount of studies have been reported in the literature concerning fires in enclosures with square and rectangular geometries, little attention has been given to investigate the fire plume flow characteristics through vertical shaft in tall buildings. The fire plume flow characteristics in tall shafts are different from room enclosure fires [11]. The stack effect is a key driving force for the movement of smoke in a tall vertical shaft [12]. Moreover internal fire whirls might be induced in tall shafts under fires [13,14]. In a vertical shaft, flame is surrounded by side walls and these boundaries would trap the heat and restrict the air movement horizontally and generate a swirling motion [15,16]. The buoyancy driven flow and heat transfer characteristics in a tall shaft with side opening in the laminar regime have been studied numerically and reported in [17,18].

The computational study on buoyancy driven mixing between two compartments with heavier fluid over a lighter fluid separated with horizontal vent is analyzed by allowing the fluids to mix through a centrally located vent [19–22]. A density difference arises between the fluids due to temperature difference and this drives the interaction between the two chambers. The unsteady mixing phenomenon occurs between two chambers due to Rayleigh–Taylor instability and leads to the formation of thermal plumes. The flow characteristic of turbulent mixing in channels and rectangular enclosures due to Rayleigh–Taylor instability was presented in the literature [23–25].

The fire transport phenomena involve complex processes and buoyancy induced turbulent mixing is of great importance since it is one of the key processes for the growth and spread of fire [26,27]. Moreover the selection of turbulence model governs the accuracy for predicting the critical parameters such as fire plume height and width, gas temperature, velocity field, smoke filling time and volumetric flow rates through vents [28,29]. The turbulent penetrative and recirculating flow in vented compartment with a fire source are numerically analyzed using the low Reynolds number $K-\varepsilon$ model of Lam–Bremhorst [30]. Similar numerical studies on buoyancy induced turbulent flows using the Lam–Bremhorst model were reported [31,32] and found that this model has higher capability for predicting the buoyancy induced turbulent quantities reasonably well in regions near the wall and away from the walls. Extensive numerical investigations were performed by Kenjeres et al. [33–36] on turbulent natural and mixed convection problems using transient Reynolds-averaged-Navier–Stokes (TRANS) with the turbulence model and their results showed good agreement with LES, DNS and experimental results. A direct numerical simulation method in predicting the buoyancy induced flow characteristics was investigated and reported in [37–44].

A tall building consists of open vertical shafts with multiple openings at different levels in vertical and horizontal partitions. In summary, the earlier works reported in the literature have not investigated the combined effects of flow through horizontal and vertical vents in partitioned enclosure connected to neighboring tall vertical open shaft. Hence, not much work has been done on the combined counter current exchange flow through the multiple vertical and horizontal openings between compartment and vertical shaft. In the case of tall buildings the smoke movement through vertical shafts generates internal fire whirls and is significantly important because the buoyancy force that draws smoke upward inside these shafts encounters little resistance as the smoke rises to upper floors. Moreover such transport phenomena pose a threat to human life at remote locations from the fire. This has been the motivation for present investigation.

Here the computational study is performed on the fire induced turbulent flow characteristics in a tall open vertical shaft connected to an adjacent rectangular compartment which is the integral part of high-rise buildings. The stream function and vorticity formulation with the $K–\varepsilon$ low Reynolds number turbulence model of Lam–Bremhorst are used to solve the governing equations. Results are reported with different Grashof numbers. The effects of vent size and combined effects of flow through multiple vertical and ceiling vents are reported. The present results are matching very well with the numerical and experimental results available in the literature.