International Journal of Thermal Sciences 90 (2015) 135–149

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Airflow induced by a room fire: Effect of roof shape and source location

Ons Tlili [a,*], Hatem Mhiri [a], Philippe Bournot [b]

[a] UTTPI, Ecole Nationale d'Ingénieurs de Monastir, Route de Ouardanine, 5000 Monastir, Tunisia
[b] IUSTI, Technopôle de Château-Gombert, 5 rue Enrico Fermi, 13 013 Marseille, France

ARTICLE INFO

Article history:
Received 6 March 2014
Received in revised form
10 December 2014
Accepted 12 December 2014
Available online 7 January 2015

Keywords:
CFD modeling of airflow
Fire source in enclosure
Roof shape effect
Doorway

ABSTRACT

A computational study of fire induced airflow in an enclosure is presented. The fire is modeled by a volumetric heat source centrally located in a rectangular room with a door which provides natural ventilation. The first part of this work aims at evaluating the performance of three turbulence models to predict the airflow pattern inside the fire room. Numerical results are validated with available experimental data and it is concluded that the standard k–ε turbulence model coupled with an improved treatment of the wall functions namely the 'enhanced-wall treatment' gives the best compromise between accuracy of results and computation time. This latter is therefore applied in the second part of this work which includes an analysis of the impact of the roof shape on the hot gases evacuation process for several heat source locations inside the room. Two typical roof shapes are tested, a pyramidal roof and a domed one. Comparison of these results with those obtained with the basic model (flat roof) allowed the assessment of the major influence of the two critical parameters studied on the temperature and velocity distributions inside the fire room and the calculation of the design parameters such as the neutral plane height and mass flow rates at the opening for each fire scenario.

1. Introduction

Fires are destructive and have unpredictable consequences; the main risk factors being caused by the spread of the fire fumes (toxic gas, soot and heat). An analysis of the combustion process and the smoke spread is thus required in order to provide the necessary tools to assess risks related to the fire.

Several investigations [1–3] relating to the design of fire safety systems have been conducted, in order to study the flow that occurs after the fire ignition and to prevent the smoke layer from descending to an occupied zone.

Thus, the quantification of the smoke layer properties as well as the mass flow rates through the openings has been a prime aim of physicists and engineers [4,5].

Many experimental studies examining the flow induced by fire in a compartment have been carried out: a detailed study was performed by Steckler et al. [6] who conducted a series of full-scale experiments of the flow induced by fire in a compartment. Experimental results about thermal and dynamical characteristics of the flow (temperature, velocity and mass flow rates) were obtained for different heat source locations inside the fire room. These results were necessary to validate various Computational Fluid Mechanics codes [7] and to test geometrical parameters effects on the flow field.

Although experimental studies on fire scenarios are of a great interest and accuracy in the prediction of the fire induced flow, they are not always easy to perform because of the devastating consequences that can result. On the other hand, the complexity of indoor airflow makes experimental investigation extremely difficult and expensive. This is why we often resort to the CFD modeling.

As the airflow induced by thermal sources inside an enclosure is characterized as buoyant and turbulent, CFD modeling is considered the most suitable tool for reliable airflow simulations. It is also useful for detecting the detailed effects of different parameters (geometrical, thermal ...) acting on the phenomenon [8–11].

The CFD modeling is based on the fundamental local conservation laws for physical quantities, such as mass, momentum, energy and chemical species concentrations. The spatial and temporal resolution of these equations provides detailed information of the flow structure, which explains why this approach is considered a

* Corresponding author. Tel.: +216 23296654.
E-mail address: tlili_ons@hotmail.fr (O. Tlili).

http://dx.doi.org/10.1016/j.ijthermalsci.2014.12.003
1290-0729/

Nomenclature

CFD	Computational Fluid Dynamics
VHS	Volumetric Heat Source
g	gravitational acceleration (m s^{-2})
H	opening height (m)
k	turbulence kinetic energy (m^2 s^{-2})
$\dot{m}_a$	mass-inflow rate (kg s^{-1})
$\dot{m}_g$	mass-outflow rate (kg s^{-1})
N	neutral plane height (m)
P	pressure (Pa)
S	Source term (J)
t	time (s)
T	temperature (K)
U	mean velocity (m s^{-1})
x,y,z	Cartesian co-ordinates

Greek symbols

β	thermal expansion coefficient (1/K)
ε	turbulence dissipation rate ($m^2.s^3$)
λ	thermal conductivity (W m K^{-1})
μ	dynamic viscosity (PI)
ρ	density (kg m^{-3})
ν	kinematic viscosity (m^2 s^{-1})
σ	turbulence Prandtl number
α	thermal diffusivity (m^2 s^{-1})

Subscripts

i,j	directions
lam	laminar
t	turbulent
k	kinetic energy of turbulence
ε	dissipation rate of turbulence

general and accurate tool to deal with compartment fires phenomenon.

The use of the CFD modeling varies from the validation of new computational methods or turbulence models to a parametric study in complex geometries. For example, R. Hasiba et al. [12] used their own experimental results to conduct a CFD parametric study using the CFX code in order to evaluate the effect of the geometrical and thermal parameters of the heat source on the smoke spread.

A.A. Peppes et al. [13] worked on a more complex problem, modeling the airflow through a stairwell connecting two floors of an industrial building.

The most challenging task in the use of CFD codes is the choice of the appropriate approach (DNS, RANS or LES) and the selection of a suitable turbulence model to represent the phenomenon of interest. H. Xue et al. [14] conducted a comparative study between different combustion models in typical cases of enclosure fire simulation and examined the performance of three models for different airflow problems (VHS model, the eddy break-up model and the pre-PDF model) in order to find an adequate turbulent combustion model for use in enclosure fires. It was shown that all the tested models produce satisfactory results in the prediction of general flow pattern; however, each model is suitable for a specified fire scenario and geometry (Tunnel, compartment, Atrium …).

A similar study aiming at assessing the performance of various turbulence models in predicting airflow in an enclosed environment has been conducted by Z. Zhai et al. [15] and Z. Zhang et al. [16], they firstly advanced various high-Reynolds number turbulence models that are commonly used in enclosure fire modeling, and afterward evaluated the generality and robustness of these turbulence models for various airflow scenarios by using experimental data from the literature. As a conclusion, they reach the fact that, among the high-Reynolds number turbulence models used both the RNG k–ε and the standard k–ε with standard wall functions are widely used for airflow numerical studies because these models provide acceptable results with good computational economy. Although the second moment closure model RSM (Reynolds Stress Model), can capture some flow details that cannot be modeled by the eddy viscosity models, its biggest drawback is the long computation time.

Following the choice of an adequate turbulence model for performing a numerical simulation, the next step is to detect the effect of geometrical parameters on the airflow distribution, as the numerical study of Sherman C.P. Cheung et al. [17] who investigated the influence of the door gap sizes on the smoke spread in an enclosure fire.

H-Jin Park et al. [18] also performed a two-dimensional computational simulation to examine the impact of the heat source location on the ventilation systems.

As it can be seen, parametric studies generally inspect the effect of the heat source locations on the enclosure (center, corner, and sidewall), the source size, and the geometry of the compartment. We therefore chose in this work to focus on a parameter that has not been treated previously and which has a great influence on the airflow distribution, this parameter is the roof shape.

Referring to the experimental results found in Ref. [6], we performed in this paper, a numerical simulation of the airflow induced by a heat source in a compartment. The fire is represented by a Volumetric Heat Source placed in the centre of the room (VHS approach).

The VHS modeling is an approach of combustion modeling which does not take chemical reaction into account; instead, it sets the heat release rate equivalent to that of the assumed fire. This approach can adequately describe the thermal and dynamic fields of the fire induced flows ensuring a good accuracy of the results with shorter computation time compared to various combustion models. This advantage prompted us to use it in our work as the case of several previous researches [14,15,16, and 19].

Based on the results of G.M. Stavrakakis et al. [19] who have numerically investigated the airflow occurring in this geometry using two high-Reynolds number turbulence models (traditional k–ε model and RNG k–ε model), the main weakness of these models is their failure to predict flows in the vicinity of the sidewall. Thus, in order to eliminate these inaccuracies we investigate in the present work in the first case the effectiveness of two low-Reynolds number turbulence models in the induced airflow predictions that have not been treated previously: The S-A model and the standard k–ω model modified to account buoyancy effects.

As the major drawback of this class of turbulence models is essentially the computation time, we compared in the second case, the results of these two turbulence models with those obtained with the standard k–ε model. This former is not used herein in its traditional form, but rather modified with a near-wall model approach, namely ‘Enhanced Wall Treatment’.

It is concluded that the results obtained by the ‘Enhanced Wall Treatment’ applied to the standard k–ε model, together with the VHS approach led to a good agreement with the experimental measurements [6] and provided a plausible prediction of the temperature and velocity distributions. The accuracy of this turbulence model has also been proven in the calculation of the mass flow rates at the opening and in the prediction of the neutral plane height.