International Journal of Heat and Mass Transfer 81 (2015) 578–590

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical simulations of fire-induced doorway flows in a small scale enclosure

S. Suard *, A. Koched, H. Pretrel, L. Audouin

Institut de Radioprotection et de Sûreté Nucléaire (IRSN), ETIC Laboratory, BP 3, 13115 St Paul-Lez-Durance Cedex, France

ARTICLE INFO

Article history:
Received 11 June 2014
Received in revised form 28 October 2014
Accepted 29 October 2014
Available online 16 November 2014

Keywords:
CFD model
ISIS
Fire modeling
Doorway flows
SPIV

ABSTRACT

Field simulation results of a fire, centered in a small scale enclosure (1.04 m^3) with an open doorway, performed with the CFD computer code ISIS, are presented. Three heat release rates of 10.6, 15.5 and 21.7 kW, provided with a propane gas burner, are numerically studied and compared to experimental results. Turbulence and soot modeling are first validated by simulating the gas burner fire in an open atmosphere for the three fire powers. Due to the complex role of buoyancy in production of turbulence inside a pool fire plume, anisotropic modeling, through the generalized gradient diffusion hypothesis, is considered in the standard k–ε model. The comparisons between experimental measurements and numerical simulation results, for the enclosure fire, concern temperature and velocity profiles at the doorway and temperature profiles inside the enclosure. Velocity measurements at the open doorway are performed using a stereoscopic particle image velocimetry (SPIV) technique, allowing a full comparison with computational fluid dynamics (CFD) results. For the three heat release rates, the simulation results agree well with experimental measurements. General flow patterns, provided by CFD simulations, are reported for the highest fire power and supply useful information for understanding enclosure fires.

1. Introduction

Modeling confined fire inducing natural convective flow through an opening is still a challenge for the fire community. Usually, a fire scenario involves several complex phenomena such as phase-change process for liquid/solid materials and chemical processes for the gaseous phase. This later brings strongly coupled physics which are non-premixed turbulent combustion, soot production, radiative transfer, wall heat transfer and turbulent motion of smoke inside the enclosure and through the open doorway. Two types of fire models are commonly used to simulate such complex phenomena: zone models and field models. In the field of fire protection engineering, zone models [1,2] are frequently used due to their ease of use and to their short computing time, which make them suitable for studying fire hazards or designing protection systems. However, the two-zone assumption cannot be satisfied for each fire scenarios, especially in the case of important ratios between the fire heat release rate and the volume of the enclosure. Indeed, for these configurations, the presence of two distinguish layers is not completely observed and the use of a field or CFD model is preferable even despite the long computing time. Nevertheless, validation of such models is a difficult task. Several validation studies have considered the experiments done by Steckler et al. [3–5] as a suitable test case for field model validation. Even though these fire experiments were well documented, the bidirectional probes, used to measure the velocity in these studies, are intrusive. Also, they do not allow a determination of the different velocity components as they only measure an average velocity magnitude over their inlet surface. Indeed, some recent research works [6,7] have shown that velocity measurements using bi-directional probes, as in Steckler's experiments [3,4], may be overestimated in comparison with non-intrusive laser techniques as particle image velocimetry (PIV). This optical measurement method, detailed in [8,9], allows a complete description of turbulent flows by measuring instantaneous velocity fields with a high spatial and temporal resolution. In the field of fire science, there were some attempts to use this approach [6,7,9–11] but no formal comparisons with numerical results have been performed previously and it is one of the objectives of the present study.

The literature about gas burner fire inducing flows through a doorway is abundant but surprisingly, only few CFD simulations, including a comprehensive modeling of an enclosure fire and a complete validation with measurements inside and outside the compartment, have been performed. Apart from general works about CFD fire modeling [12–15] and some specific research topics

* Corresponding author. Fax: +33 4 42 19 91 61.
E-mail address: sylvain.suard@irsn.fr (S. Suard).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.10.069
0017-9310/

Nomenclature

C_p	specific heat capacity at constant pressure (J/(kg K))
C_s	soot formation parameter (kg/(N m s))
G	average incident radiation (W/m^2)
G_k	turbulent production due to buoyancy (kg/(m s^3))
$\mathbf{g}$	gravitational acceleration (m/s^2)
h	enthalpy (J/kg)
$\mathbf{I}$	identity tensor
I	radiation intensity (W/(m^2 sr))
I_t	turbulent intensity (–)
k	turbulent kinetic energy (m^2/s^2)
l	Inlet turbulent characteristic length scale (m)
$\dot{m}$	mass flow rate, (kg/s)
n	exponent for soot formation
$\mathbf{n}$	unit normal vector to a surface
P_k	turbulent production term (kg/(m s^3))
P_{th}	thermodynamical pressure (Pa)
p	dynamic pressure (Pa)
p_F	fuel partial pressure (Pa)
q_r	radiative heat flux (W/m^2)
$\mathcal{R}$	universal gas constant (J/(mole K))
s	mass stoichiometric ratio (–)
S	Surface (m^2)
$\mathbf{S}$	strain rate tensor (1/s)
T	temperature (K)
u_m	mean velocity (m/s)
u_τ	friction velocity (m/s)
$\mathbf{v}$	velocity vector (m/s)
W	molecular weight (kg/mole)
X	mole fraction (–)
Y	mass fraction (–)
z	mixture fraction (–)

Greek symbols

$\Delta h^0_{f,k}$	enthalpy of formation of species k at 298.15 K, (J/kg)
δ	normal distance from the wall (m)
ε	turbulent dissipation rate (m^2/s^3)
ϵ	emissivity (–)
ζ	constant of the wall-law model
κ	absorption coefficient (1/m)
λ	conductivity (W/(m K))
μ	dynamic viscosity (Pa s)
ϕ	variable
ψ	fuel/air equivalence ratio (–)
ρ	density (kg/m^3)
$\boldsymbol{\sigma}$	stress tensor (N/m^2)
σ	Stefan–Boltzmann constant (W/(m^2 K^4))
σ_ϕ	turbulent Prandtl/Schmidt number for variable ϕ
$\boldsymbol{\tau}$	Reynolds stress tensor (N/m^2)
τ_w	wall shear stress (N/m^2)
ω	solid angle (sr)

Subscripts

∞	ambient condition
b	buoyancy effect; Black body
e	effective
F	fuel
f	fluid
g	gas
lam	laminar
O	oxygen
r	radiative
s	soot
t	turbulent
w	wall surface

about the effect of soot [16], radiation [17–19] and turbulence modeling [20–23], the main studies carried out on enclosure fires can be classified into two categories, dependent upon the combustion process modeling. The first one considers the fire as a volumetric heat source (VHS) and no combustion model is taken into account. As a result, gas species, including combustion products, cannot be determined and, this modeling, is commonly used to obtain a rough estimation of the hot gas layer temperature and of the upper layer height. This approach is not detailed in the following but some of major works can be found in [24–27]. The second category of these numerical studies deals with the combustion process using a turbulent formulation of Navier–Stokes equations and a simplified global one-step reactive scheme. As a brief literature review, the most relevant and recent contributions of these studies, which present a validation of their results with experimental measurements, are given in Table 1. From this overview, several findings and observations have been made and have motivated the present work which is focused on a comprehensive modeling and numerical validation of fire-induced doorway flows:

1. By bringing these studies according to the volume of the fire compartment, it results that the majority of them concern large scale compartments with a volume superior to 10 m^3. Only two numerical studies of very large scale (VLS) and medium scale (MS) enclosures have been identified.
2. Through the ratio of the heat release rate to the compartment volume, which represent a global volumetric firepower for the whole compartment, Table 1 indicates that the majority of large scale (LS) simulations involve moderate ratio, less than 10 kW/m^3. The VLS studies show very disparate volumetric firepowers as [28] simulates fires under ventilation-controlled condition whereas [29] investigates doorway flows induced by a 100 kW fire in highly over-ventilated combustion. A small volumetric firepower represents a suitable configuration for CFD models as this fire scenario automatically induces a lower upper layer temperature and low temperature gradients between the compartment zones.
3. The mesh resolution of the simulations obviously depends of the size of the computational domain and is respectively about 2–5, 5 and 10 cm for MS, LS and VLS enclosures. A mesh convergence is, theoretically, hard to achieve mainly because of the computing power required for such simulations, but also because of the modeling used by the different field fire models. Nevertheless, a large number of local phenomena, which potentially have a significant impact on the flow motion, cannot be simulated properly with such grids.
4. For some experimental configurations with large openings, allowing a huge part of heat to be convected out of enclosure [30], the adiabatic condition for walls gives proper results. In contrast, fire scenarios with a high volumetric firepower or without insulated walls, require to solve the conductive heat transfer through the walls with an appropriate convective flux modeling. In this case, the convective coefficient should not be fixed, a priori, to a constant value but should take into