Fire Safety Journal 46 (2011) 186–193

Contents lists available at ScienceDirect

Fire Safety Journal

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

Verification of the accuracy of CFD simulations in small-scale tunnel and atrium fire configurations

Nele Tilley*, Pieter Rauwoens, Bart Merci

Ghent University, Department of Flow, Heat and Combustion Mechanics, Sint-Pietersnieuwstraat 41, B-9000 Ghent, Belgium

ARTICLE INFO

Article history:
Received 1 June 2010
Received in revised form
17 January 2011
Accepted 19 January 2011
Available online 15 February 2011

Keywords:
Smoke control
CFD
Numerical experiments
Atria
Tunnels

ABSTRACT

In preparation for the use of computational fluid dynamics (CFD) simulation results as 'numerical experiments' in fire research, the agreement with experimental data for two different small-scale set-ups is discussed. The first configuration concerns the position of smoke-free height in case of fire with vertical ventilation in an atrium. The second set-up deals with the critical velocity for smoke backlayering in case of fire in a horizontally ventilated tunnel. An *N*-percent rule is introduced for the determination of the presence of smoke in the simulation results, based on the local temperature rise. The CFD package FDS is used for the numerical simulations. The paper does not scrutinize the detailed accuracy of the results, as this is hardly possible with any state-of-the-art experimental data at hand. Rather, the global accuracy is discussed with current numerical implementation and models in FDS, considering continuous evolution over different version releases with time. The agreement between the experiments and numerical simulations is very promising. Even when quantitative agreement with experimental data is not perfect, the trends are very well reproduced in the simulations. While much additional work is required, both in CFD as in 'real' experiments, the results are encouraging for the potential of state-of-the-art CFD to be used as numerical experiments.

1. Introduction

Fire safety standards for buildings have long time been based on prescriptive rules. However, there is a world wide evolution towards performance-based design, particularly for large, complex buildings. The question can indeed be raised whether current standards still prevail for complex buildings and modern architecture. Supportive insight in the (lack of) fire safety in a design fire scenario can be provided by the application of computational fluid dynamics (CFD), which can be performed for a specific configuration. However, one step beyond is to consider CFD simulations as 'numerical experiments'. Numerical simulations are relatively cheap (at least in comparison to real large-scale experiments). However, a substantial knowledge of the user is required to perform high-quality CFD simulations and careful application is mandatory. The main advantage of numerical simulations is that a significant amount of different parameters can be varied in order to study their effect. As such, this can lead to further development of fire safety standards. It is desirable to exploit this approach in fire safety research. In particular, one research objective is the improvement of calculation methods to determine the required smoke extraction rate to meet fire safety objectives (such as smoke free heights or smoke free zones) in different types of buildings; including atria and large closed car parks.

Obviously, a 'conditio sine qua non' is then that the CFD simulation results are reliable, i.e. of sufficient accuracy in 'blind' circumstances. Therefore, as a first step to show that CFD has the potential to be used as 'numerical experiments', two experimentally studied small-scale test cases are extensively investigated in this article. The first case concerns fires in a small-scale atrium [1]. A fire in a room, adjacent to the atrium, causes a spill plume to rise in the atrium.

The second test case is a small-scale tunnel experiment [2] with forced mechanical ventilation imposed to avoid the smoke backlayering from the fire. Note that the flow is essentially horizontal, in contrast to the atrium configuration.

For the CFD results, the simulation program Fire Dynamics Simulator (FDS, version 5) [3,4], developed by NIST, was used. However, in principle, other CFD packages could have been used as well. Indeed, it would be very valuable to repeat the study with other CFD packages, investigating their model capability, but this is considered beyond the scope of the present paper. The influence on the results of computational mesh and the thermal boundary conditions will be considered. Most importantly, though, it will be illustrated that agreement of simulation results

* Corresponding author. Tel.: +32 9 264 32 91; fax: +32 9 264 35 75.
E-mail address: Nele.Tilley@UGent.be (N. Tilley).

0379-7112/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.firesaf.2011.01.007

Nomenclature

A	area, m^2
A_c	ceiling area, m^2
A_n	area of cell n, m^2
C	coefficient, kg/(m s kW$^{1/3}$)
C_m	coefficient, m$^{4/3}$/(s kW$^{1/3}$)
c_p	heat capacity, J/kg K
C_s	Smagorinsky constant
d	backlayering distance, m
D^*	characteristic length scale, m
D_b	thickness of emerging smoke layer, m
g	gravitational acceleration, m/s^2
H	total height of atrium, m
h	convection coefficient, W/m^2 K
I_1	parameter, K m
I_2	parameter, m/K
I_b	radiation intensity source term, kW/m^2
k	cell number
k	turbulent kinetic energy, m^2/s^2
M	mass flow rate, kg/s
M_b	emerging smoke layer mass flow rate, kg/s
N	number in N-percentage rule
Q	total heat release rate of fire, kW
Q_{cond}	conductive heat loss, kW
Q_{conv}	convective heat release rate of fire, kW
Q_{loss}	heat release rate lost due to cooling, kW
Q_{rad}	radiative heat loss, kW
T	temperature, K
t	time, s
$T_{av,s}$	average smoke layer temperature, K
T_{int}	interface temperature of the smoke layer, K
T_k	temperature of cell k, K
T_l	minimum temperature on vertical line, K
T_{max}	maximum temperature, K
T_o	ambient temperature, K
t_t	turbulent time scale, s
v	velocity, m/s
v_{cr}	critical velocity m/s
v_{in}	inlet velocity, m/s
v_{out}	outlet velocity imposed at ceiling of atrium, m/s
W	width of atrium, m
x	length, m
z	height above spill edge, m
z_0	virtual origin height, m
z_{int}	interface height of the smoke layer, m
z_k	height of cell k above ground, m
Δz	vertical cell size, m
χ_r	radiative fraction
ε	turbulent dissipation, m^2/s^3
ρ_o	ambient density, kg/m^3

and experimental data is satisfactory for a wide range of tests, which is promising for the use of CFD as 'numerical experiments'.

2. Atrium

In this section, the atrium simulations are discussed. In total, 16 simulations have been performed. Four different heat release rates were studied. For each value of heat release rate, four different extraction rates are imposed. First, the set-up of the original experiments is explained. Afterwards, the numerical set-up of the simulations is discussed. The search for a reliable method to determine the smoke interface height, based on the temperature profile in the atrium, is discussed in a separate section. Finally, the numerical simulation results are presented.

2.1. Experimental set-up

In a recently published paper [1], Poreh et al. carried out a series of experiments in a small-scale atrium configuration (Fig. 1). Four different total fire heat release rates (Q_{conv}) were created in the room adjacent to the atrium. For each heat release rate, different mass flow rates of smoke (M) were mechanically extracted at the ceiling of the atrium, corresponding to a certain smoke free height above the spill edge (z_s) in the atrium. The depth (D_b) and mass flow rate (M_b) of the smoke layer, emerging from the adjacent room, were measured.

The room adjacent to the atrium has size 1.25 m $\times$ 0.9 m $\times$ 0.6 m, and the atrium itself is 2.5 m $\times$ 0.9 m $\times$ 3.6 m large.

From these experiments, Eq. (1) was deduced in [1] to calculate, for a certain heat release rate, the required smoke extraction mass flow rate, in order to maintain a specific smoke free height above the spill edge in the atrium:

$$\frac{M(z)}{Q_{conv}^{1/3}} = C(z+z_0) \quad (1)$$

with

$$C = 0.3C_m\rho_0 W^{2/3} \quad (2)$$

$$z_0 = D_b + \frac{M_b}{CQ_{conv}^{1/3}} \quad (3)$$

and C_m=0.21 for adhered spill plumes.

More recent studies of air entrainment in spill plumes in atria include [5,6]. Here, the intention is not to provide new correlations or insights. The only aim is to illustrate the quality of CFD results, in agreement with the experimental data reported in [1].

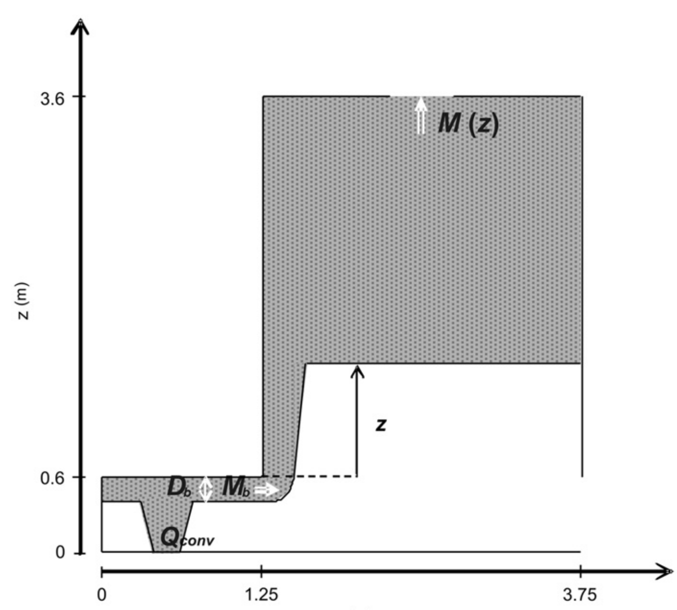

Fig. 1. Atrium configuration.