Case Studies in Thermal Engineering 4 (2014) 129–135

Contents lists available at ScienceDirect

Case Studies in Thermal Engineering

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

Numerical case studies of vertical wall fire protection using water spray

L.M. Zhao, Y.M. Zhang *, H.Z. Zhao, J. Fang, J. Qin

State Key Laboratory of Fire Science, University of Science and Technology of China, No. 96, JinZhai Road, Hefei, Anhui 230026, PR China

ARTICLE INFO

Article history:
Received 21 July 2014
Received in revised form
5 September 2014
Accepted 5 September 2014
Available online 16 September 2014

Keywords:
Numerical simulation
Fire resistance
Heat transfer
Suppression
Water spray

ABSTRACT

Studies of vertical wall fire protection are evaluated with numerical method. Typical fire cases such as heated dry wall and upward flame spread have been validated. Results predicted by simulations are found to agree with experiment results. The combustion behavior and flame development of vertical polymethylmethacrylate slabs with different water flow rates are explored and discussed. Water spray is found to be capable of strengthening the fire resistance of combustible even under high heat flux radiation. Provided result and data are expected to provide reference for fire protection methods design and development of modern buildings.

1. Introduction

High-rise building fires are now becoming one of the most disastrous accidents in modern cities [1]. Fires can be caused by accident or lack of preventive measures [2]. A remarkable feature of high-rise building fire is the high spread rate caused by upward movement of hot smoke and chimney effect caused by lift shaft or other vertically interconnected regions [3,4]. Discussions involving essential features of upward flame spread can be found in many studies [5–7].

Many fire protection technologies have been developed. Sprinklers, for example, have been proven to be one of the most efficient fire protection methods [8,9]. Computational fluid dynamic (CFD) methods have gained a lot of attention in fire modeling and fire protection system design in the past several decades. One of the topics is the interaction between water droplet and fire plume. Nam first developed a simplified model to simulate the sprinkler spray using droplet trajectories and proved its capability of predicting the actual delivered density (ADD) [10]. The complicated physical and chemical process of fire suppression was described in more detail in reference [11]. Results proved that numerical simulation approach which took the effects of momentum exchange, heat and mass transfer as well as chemical reaction into consideration had the capability to capture the interactions between water spray and fire plume. With the advance of numerical technique, more research methods like direct solution of ordinary differential equations [12] or large eddy simulation (LES) [13] were introduced into spray and fire modeling. This further proved that momentum of droplet played a key role in the interaction. Different fire phenomenon, such as wood fire, compartment fire and polymer fire, were also involved. Numerical study of wood fire extinguished by water sprinkler was conducted [14]. Lagrangian particle tracking procedure, coupled with gas

* Corresponding author. Tel.: +86 551 63606457; fax: +86 551 63601669.
E-mail address: drmingzh@163.com (Y.M. Zhang).

http://dx.doi.org/10.1016/j.csite.2014.09.002
2214-157X/

flow model, started to be used in sprinkler modeling. Extinguishment time predicted by numerical model was reasonably close to those measured from experiments. Water-mist suppression of compartment fires and detailed discussion on the physical process during suppression was presented by Prasad [15]. In his work, an advanced computational tool based on multi-block Chimera technique was developed. The gas phase and water-mist were described by equations of the Eulerian form. Simulation results improved understanding of various physical processes that took place during water-mist suppression of fires in large enclosures. Fundamental study on cooling effectiveness of a single water drop impinging on a hot surface was carried out by Pasandideh-Fard [16]. A Volume-of-Fluid (VOF) method was used to model the fluid mechanics and heat transfer during droplet impact. Observations showed that impact velocity had a weak effect on substrate temperature variation and heat flux. Original work on heat flux characteristic of spray wall impingement was presented in references [17,18]. Spray cooling mechanism was discussed and its capability of removing heat fluxes was further proved. Especially for intermittent spray, duty cycle was identified as the dominant parameter that affected the heat transfer efficiency. This also provides important theoretical basis for present work. Besides the discussion of interaction between flame and water spray, flame spread is also an important area in fire modeling. One of the topics is upward flame spread. This may contain co-flow flame spread [19] or heat process [20] over combustible slabs. In reference [21], finite difference numerical model was used to describe one-dimension upward flame spread. All these models provided solutions in good agreement with experimental data. Other numerical studies of upward flame spread focus on flame height correlation, further reference can be found in references [22–25].

Generally, when designing a new spray cooling system, it is necessary to make quick estimate of heat transfer during droplet impact. Present work employs numerical method and aims to validate fire resistance performance of vertical wall which is protected with water spray. Description of fundamental cases is presented first, followed by simulation results and discussion of the typical fire conditions. Based on the numerical model and simulation results, it is confirmed that it gives good estimate of fire resistance effect by comparing its predictions with those from other research.

2. Mathematical model

Fire dynamics simulator (FDS) developed by National Institute of Standards and Technology (NIST) is a popular and mature solution for computational fluid dynamic modeling. It is used in this paper to carry out the full-scale numerical simulation of typical fire cases.

Turbulent model used in this paper is LES. LES uses a space approximation method and solves directly the large-scale eddies. To justify the computational convergence, the Courant–Friedrichs–Lewy (CFL) criterion is used along with the setting of the time step. The initial time step is set automatically in FDS based on the size of a grid cell divided by the characteristic velocity of the flow. During the calculation, the time step is varied and constrained by the convective and diffusive transport speeds to ensure that the CFL condition is satisfied at each time step.

3. Typical fire cases

3.1. Dry boundary condition verification

Combustible heating by heat flux is common in building fire cases. Validation of this heat transfer process can help to verify capability of the model in dry boundary modeling. Experiment that simulated vertical wall heat transfer was designed and carried out by Meredith [26]. The condition is set in accordance with the experiment in which a 61 cm wide, 91 cm high, and 1.9 mm thick stainless steel plate with a satin finish is used. The external heat flux is kept at 6.03 kW/m^2, 8.27 kW/m^2, 16.4 kW/m^2 and 33.2 kW/m^2. The plate is heated under the external heat flux and temperature rise of the plate surface is measured and recorded.

The validation simulation of the dry boundary condition is performed mainly to verify the heat transfer model. The dimension of the computational volume is 0.92 m $\times$ 0.6 m $\times$ 0.045 m. Four groups of uniform mesh with different resolution are utilized and results are shown in Fig. 1(a). The lines are simulation results and square dots are data reproduced from reference [26]. It is clear that results predicted by the one dimension heat transfer model agree well with experiment data. And the current resolution of the uniform grid is fine enough to guarantee the computation accuracy. Then the uniform grid spacing of following simulation is kept between 10 and 15 mm in order to get a balance of calculation accuracy and efficiency.

Fig. 1(b) shows the temperature of the vertical wall surface under four different heat fluxes. It is clear that the temperature tendency predicted by the model agrees well with experiment result. It increases at first and then trends to stabilize. It also shows that the temperature increases more rapidly and the final temperature is higher under higher heat flux condition. In addition, accuracy of the findings is examined. The error is calculated as: (simulation result − experiment result)/experiment result. The maximum error is found to be 7.95% at 100 s under heat flux 33.2 kW/m^2, while most of the rest are no more than $\pm 1\%$. This further explains the agreement and proves the model's capability of predicting the temperature of heated wall surface.