Experimental and Numerical Study of Fire Suppression Performance of Ultral-Fine Water Mist in a Confined Space

LIANG Tian-shui[a,b], Liu Zhong-lin[a], XIAO Xiu-kun[c], LUO Siu-ming[b], LIAO Guang-xuan[c] , ZHONG Wei [a], *

[a]School of Chemical Engineering and Energy, Zhengzhou University, Zhengzhou 450001, China
[b]Department of Building and Construction, City University of Hong Kong, Kowloon, Hong Kong, SAR China
[c] Institute of Industry Technology of Guangzhou and Chinese Academy of Science, Guangzhou511458, China

Abstract

Ultra-fine water mist (UFM) is a clean agent with a gas like property. Fire suppression experiemt was conducted in a confined space. Numerical simulation using DPM model and dense gas model were carried out to compare with experiment. Simulaition discovered that dense gas model is more suitable for predicting UFM transportation and flow behavior compared with DPM model. Both experiment and simulation found that fire of a larger size is easier to extinguish in a compartment space, which is because fire sources can promote transportation of UFM in a compartment. Fire location also affects the fire suppression performance by promoting or suppressing the transportation of UFM.

© 2013 The Authors. Published by Elsevier Ltd. Open access under CC BY-NC-ND license.
Selection and peer-review under responsibility of School of Engineering of Sun Yat-sen University
.
Keywords: Ultral-Fine Water Mist; Fire suppression; Flow behavior

1. Introduction

Ultra-fine water mist (UFM), a clean agent with a gas like property, which is thought to be one of substitutes for halons agent [1, 2]. The recognizing of flow behavior of UFM is very necessary in fire protection. The flow behavior related to the design of UFM system, the installation of pipe line and the location of the nozzles of UFM. Adiga [3, 4] has conducted experimental and numerical study on extinguishment of cable bundle fires with ultra fine water mist. CFD method was used to predicate the time for reaching extinction concentrations. Fumiaki [5] analyzed the flow behavior around flame in a cylinder cupburner. While more complex scenarios which concern engineering practice need to be studied further, such as the obstruction in applied fields, the location of fire, fire size and the spatial shape of the applied fields.

The studies in this paper will study the behavior of UFM. Two kind spatial shapes of applied fields were concerned, which were a confined compartment and a small scale tunnel. Experiment and numerical simulation were both conducted. In numerical simulation, two kinds of model were used for simulating the UFM; one is the Discrete Phase Model (DPM), the other is density gas model. The obstruction, fire location and fire size were concerned in the study as well.

2. Experiment setup

The compartment was 1×1×1m, as shown in Fig. 1. Two circular inlets of ultra-fine mist located at xyz (0.22,0.72,0) and (0.37,0.72,0) as shown in Fig. 2. The central of the rectangle outlet of the compartment located at (0.3,0.72,1) with 0.18m

---

* Corresponding author. Tel.: +86-371-56763348, fax: +86-371-67781801
E-mail address: zhongwei@zzu.edu.cn.

length (x direction) ×0.15m width (y direction). This layout is shown in Fig. 2. The diameter of the two circular inlets was 110mm. To ensure most of UFM generated can travel to the fire location, the flow rate was 4.75L/s based on numbers of experiments. The UFM concentration was regulated by controlling the number of atomizers. The amount of UFM produced by one atomizer is 5g/Min. The total numbers of atomizer were 50. One atomization unit contained ten atomizers. Cutting off the power of one atomization unit can shut off it.

In the scenarios of extinguishing fires, the fuel pan was placed at location_1 or location_2 as shown in Fig. 2. The size of the fuel pan included 100mm, 150mm or 185mm. After the fire burned 40s, the UFM system activated. A DV outside the observation door was used to record the extinguishing time.
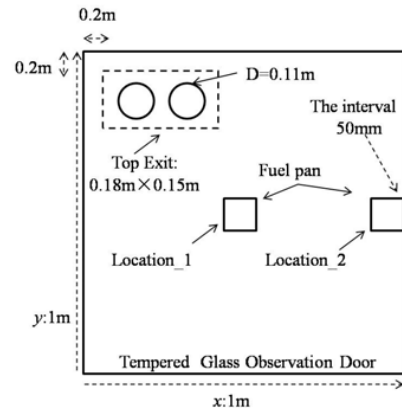

Fig. 1 The picture of the confined compartment (1m×1m×1m)

Fig. 2 Layout of the confined compartment test scenario

3. Numerical simulation

In numerical simulation, the k–ε turbulence model was used to simulate thermal plume and the air flow. The volumetric heat source model without chemistry reaction and radiation was adopted to generate thermal flame to simulate the interaction of fire with UFM. UFM inlet and air flow was imported through the same surface of which the boundary condition was velocity inlet. In simulation, to avoid the generating of unstructured grid this will lead to the increase of the total number of mesh, the circular inlet was simplified as a square one with the same sectional area of 0.1×0.1m, as shown in Fig. 3.

To simulate UFM, two kinds of model were used to compare with experiment. More information about these two models, please see the following section 3.1 and section 3.2.

The geometry and mesh of the compartment, please see the following section 3.3.

3.1. Fluent discrete phase model: Euler-Lagrange approach

The Lagrangian frame treats the UFM as the discrete phase and the fluid phase as the carrier phase using an Eulerian scheme. The fluid phase is treated as a continuous phase by solving the time-averaged N-S equations, while the particle phase is solved by tracking droplets which pass through the calculated flow field. The forces (drag force of hydrodynamic and the force of gravity) acting on particles was calculated to predict the trajectories of individual particle. In the Lagrangian particle tracking method, the interactions of particle phase with fluid phase are solved by iteratively calculating the fields at predefined intervals. The random walk tracking approach about discrete phase was activated to predict the influenced of turbulence on particle trajectories. The detail information is available in Fluent Manual [6].

The boundary condition of DPM needs to be defined. Reflect boundary condition mean that the particle may be rebounded by an inelastic or elastic collision. Trap boundary condition indicates that the particle may be trapped at the wall. For a trap boundary condition, nonvolatile material will be lost from the calculation at the point of impact with the boundary and volatile material present in the particle or droplet is released to the vapor phase at this point. Escape boundary condition means that the particle is lost from the calculation at the point where it impacts the boundary [7]. For an inert particle, escape boundary condition is the same with trap boundary condition. As a matter of fact, UFM is absorbed by the wall boundary. In the study the Discrete Phase Model (DPM) Boundary Conditions is defined as trap type.

The interaction of turbulence with particle phase needs to be modeled. In the study stochastic tracking model was activated to simulate turbulent dispersion of particles. In a turbulent flow, the trajectories of particles is predict by using the mean fluid phase velocity, u, in the force balance equations

$$\frac{du_p}{d_t} = F_D(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} + F_x$$

(1)

Instantaneous value of the fluctuating gas flow velocity $u = \bar{u} + u'(t)$ should be included to predict the dispersion of the particles due to turbulence as well. The random effect of turbulence on the particle transportation is considered by calculating the trajectory in this manner for a sufficient number of representative particles (predefined as 0.15). Instantaneous gas velocity was determined by using a stochastic approach. In the stochastic approach, the random velocity term ($u'(t)$) simplified as a discrete piecewise constant functions of time. Their fluctuation value is the same value over a time interval which was calculated by the characteristic lifetime of the turbulence eddies.

3.2. Dense gas approach

For simplicity, UFM also can be treated as a dense gaseous species (with molecular weight M=24370 kg/kg-mol) that is different from H2O. The molecular weight was calculated using ideal gas equation with the density of water, temperature, pressure and gas constant (1000 kg/m3, 293 K, 1×105 N/m2 and 8.314 J/(K • mol), respectively). Such a treatment is reasonable for UFM (<10μm) [5,7].The species Dense Gas (DG) has the transport properties of water vapor except for the density. The density of DG is 1000 kg/m3. The DG has the thermodynamic properties of liquid water. The DG was injected at the mist inlet with air flow as in the case of DPM.

In the simulation study, we focus on the flow behavior of UFM. The vaporization of UFM were not considered in simulation.

3.3. Geometry and mesh: the confined compartment

This layout of the confined compartment is shown in Fig. 3. The inlets were given velocity inlet conditions to simulate the flow inlet. The inlet velocity is 0.4m/s. The outlet is pressure outlet boundary condition. The others boundary were wall. The volume of the volumetric heat source is 0.1m×0.1m×0.1m. The energy power of the volumetric heat source was 5×106 w/m3. The geometry was meshed using 0.02×0.02×0.02m grid.

In simulation using DPM, the water mist was injected from this surface. The diameter of the UFM is 10μm. The total mass flow of ultra-fine mist was equally divided amongst these two rectangle mist inlet. The flux density was calculated by dividing the volume flow rate of air by the mass flow rate of UFM. The total flow rate of UFM was 0.012kg/s.

In simulation using DG model, the DG and air flow was inject from the inlet surface. The mass fraction of DG was 0.33 and the mass fraction of oxygen is 0.154, which mean the same amount of water was inject with the cases using DPM.
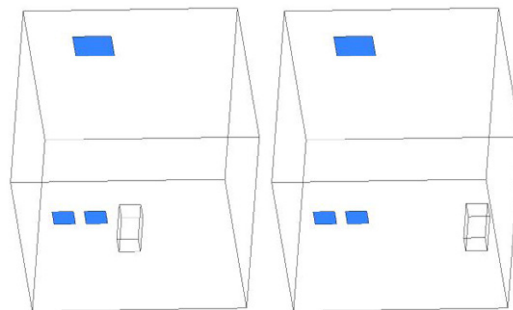

Fig. 3 Layout of a confined compartment simulation scenario (Left: fire located at the Location_1; Right: fire located at Location_2)

4. experiment and simulation results

4.1. Confined space without fire

Fig. 4 shows the filling process of UFM in the confined compartment. Fig. 5 shows the isosurface of mass fraction at the value of 0.2 or 0.118 in the simulation of using Dense Gas model. Fig. 6 shows the isosurface of particle concentration at the value of 0.306g/L in the simulation of using discrete phase model. Fig. 7 shows the contours of mass fraction in the simulation using density gas model. Fig. 8 shows the contours of mass concentration in the simulation using discrete phase model.

From Fig. 4 we can see that the filling time for UFM was approximately 100s. In the experiment, the mass concentration of UFM was not measured because ray cannot pass through the volume filled with UFM. Fig. 5 shows at 100s the minimum mass fraction of UFM in the compartment is approximately 0.118 in the simulation using Dense Gas model. Fig. 7 shows that the isosurface was a smooth curved surface. From Fig. 6 and Fig.8 we can see that the distribution of mist concentration was more non-homogeneous using DPM. From Fig. 5 and Fig. 6 we can see that the filling time of DPM was longer than the Dense Gas model.

From the comparison of Fig. 4, Fig. 5 and Fig. 6 and the comparison of of Fig. 4, Fig. 7 and Fig. 8 we can see that the results of Dense Gas model were better agree with the experiment.
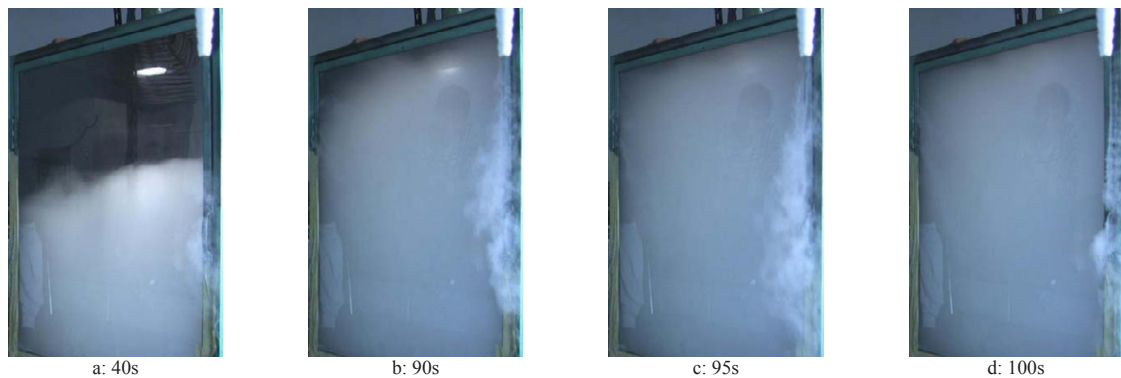

a: 40s   b: 90s   c: 95s   d: 100s

Fig. 4 pictures of the filling process with UFM in confined compartment in experiment

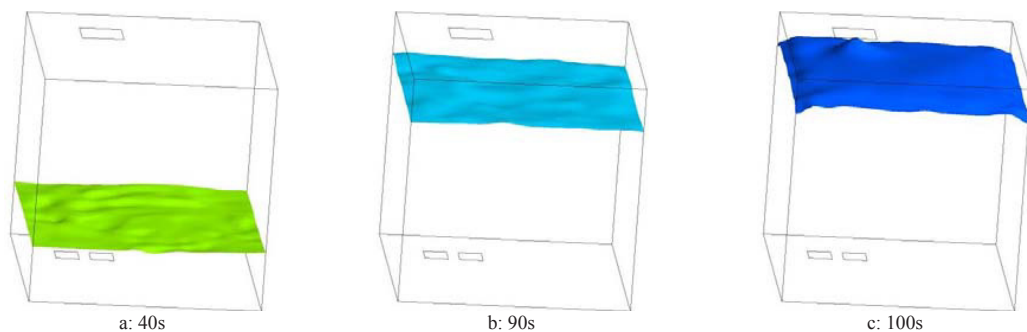a: 40s   b: 90s   c: 100s

Fig. 5 The isosurface of mass fraction (the value is 0.118) in confined compartment using density gas model (simulation results)

4.2. Comparison of flow behavior in compartment with and without fire (simulation results)

Fig. 9 shows the isosurface of mass fraction of UFM (the value is 0.2 and 0.25, respectively) in confined compartment using density gas model at simulation time 37s.

Comparing Fig. 9a, Fig. 9b and Fig. 9c, we can see that fire plume can improve the diffusion and transportation of UFM in compartment. The improvement depends on fire location. The middle location has more improvement than the side location.

Based on the simulation results, we can infer that the fire with larger power will easier to extinguish than the small one, and that the fire located at the middle will easier to extinguish than the side one.

4.3. Experiment results of extinguishing fire in confined compartment

Table 1 shows experimental results of each scenario conducted in the compartment. From Table 1 we can see that the extinguishing time decrease with the increase of the size of fuel pan, and that the fire located at the middle is easier to extinguish than the side one. Both of them agree well with the inference based on simulation results.
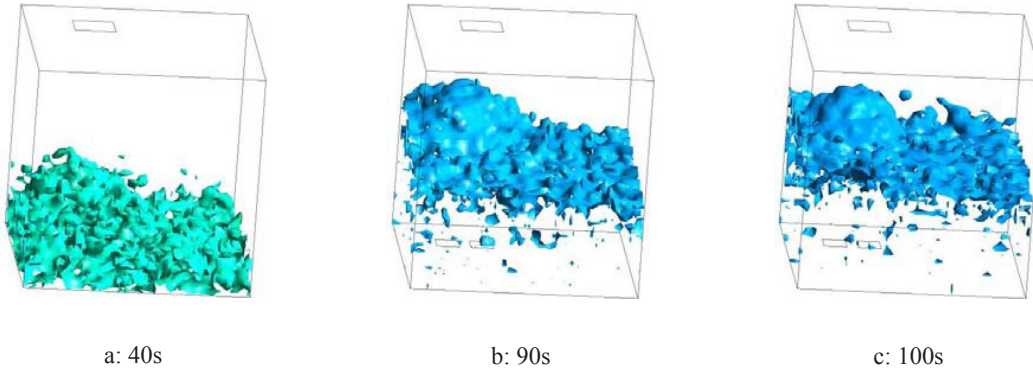

| a: 40s | b: 90s | c: 100s |

Fig. 6 The isosurface of mass concentration (the value is 0.164g/L) using discrete phase model (simulation results)
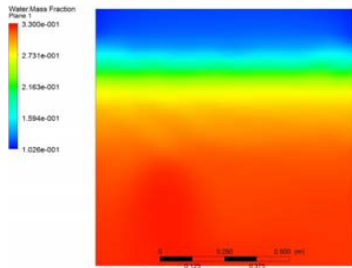

Fig. 7 The contours of mass fraction using dense gas model (simulation results)
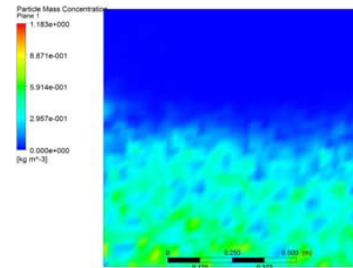
Fig. 8 The contours of mass concentration using discrete phase model (simulation results)
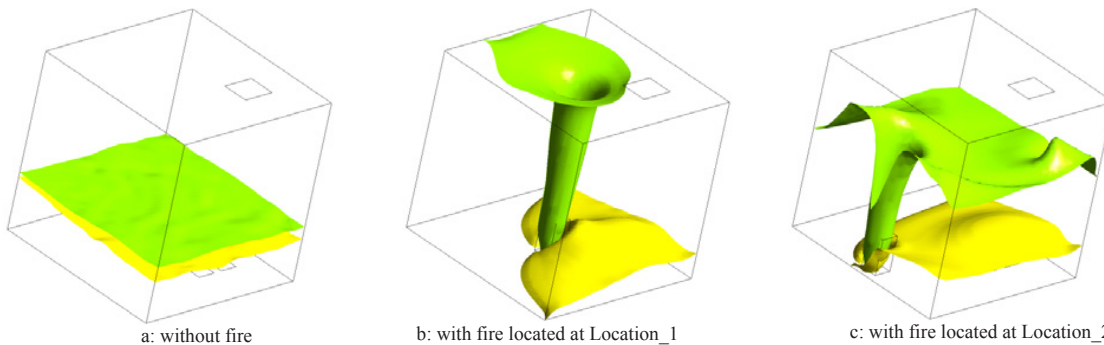

| a: without fire | b: with fire located at Location_1 | c: with fire located at Location_2 |

Fig. 9 The isosurface of mass fraction (the value is 0.2 and 0.25) in confined compartment using density gas model (simulation results at 37s)

Table 1 experimental scenarios and results in the compartment

Scenario	Mist concentration	Fire size	Fire location	Extinguishing time
1	0.877g/L	10cm	Location_1(middle)	80s
2	0.877g/L	15cm	Location_1(middle)	50s
3	0.877g/L	18.5	Location_1(middle)	29s
4	0.807g/L	18.5	Location_1(middle)	46s
5	0.807g/L	15cm	Location_1(middle)	58s
6	0.807g/L	18.5	Location_2(side)	92s
7	0.807g/L	18.5	Location_1 with 50mm height	50s
8	0.807g/L	18.5	Location_1 with 100mm height	46s

5. Conclusion

CFD simulation and experiment results on flow behavior of UFM were presented in this section. The conclusions can be summarized as follows:

1) The CFD simulations using DPM cannot simulate the transport and flow behavior of the low momentum UFM. The dense gas model showed a significant improvement in predicting the UFM transportation and flow behavior.

2) Larger size of fire is easier to extinguish in compartment space, which can promote the transportation of UFM in compartment.

Acknowledgements

References

[1]   Liang Tianshui, Lo Siuming, Liao Guangxuan. Study on Fire Extinguishing Performance of Ultra Fine Water Mi st in a Cup Burner[J]. SCIENCE CHINA Technological Sciences,2012, 55(7): 1982-1987
[2]   Fisher B T, Awtry A R, Sheinson R S,et al. Flow behavior impact on the suppression effectiveness of sub-10-lm water drops in propane/air co-flow non-premixed flames[J]. Proc Combust Inst, 2007, 31(2): 2731–2739
[3]   Adigaa K C, Hatcher R F, et al. Ultra fine water mist application in aircraft hidden fires[C], The Navy Technology Center for Safety and Survivability Chemistry Division,Naval Research Laboratory,Washington DC.14th Annual International Halon Options Technical Working Conference(HOTWC).Albuquerque:BFRL Publications,2004:125 -137.
[4]   Adigaa K C, Hatcher R F. Ronald S. Sheinson, et al. A computational and experimental study of ultra fine water mist as a total flooding agent. Fire Safety Journal 42 (2007) 150–160
[5]   Fumiaki T, Viswanath R K. Extinguishment of diffusion flames around a cylinder in a coaxial air stream with dilution or water mist. Proceedings of the Combustion Institute 32 (2009) 2615–2623
[6]   FLUENT 6.3 User's Guide. Chapter 22. Modeling Discrete Phase. Fluent Inc.
[7]   Adigaa K C, Hatcher R F, Sheinson RS, et al. A computational and experimental study of ultra fine water mist as a total flooding agent. Fire Safety Journal, 2007;42:150–160