18th Australasian Fluid Mechanics Conference
Launceston, Australia
3-7 December 2012

Characterization of a water-mist spray: Numerical modelling and experimental validation

H.M.I. Mahmud, K.A.M. Moinuddin and G.R. Thorpe

Centre for Environmental Safety and Risk Engineering
Victoria University, P.O. Box 14428, Melbourne City MC, Victoria, 8001 AUSTRALIA

Abstract

Water-mist is emerging as an effective fire suppression agent. Water-mist nozzles use a little amount of water compared to conventional sprinklers and can extinguish fires quickly with minimal damage to property. However, the performance of water-mist nozzles is greatly influenced by the behaviour of sprays they produce. Hence, the characterization of sprays is essential to predict and evaluate the behaviour of sprays produced by nozzles. A computational fluid dynamics (CFD) model is a tool for the characterization of sprays. However, it is necessary to evaluate the capability of a CFD model in predicting the behaviour of a sprays before using it for such characterization. Therefore, the purpose of this study is to investigate the capability of a numerical model to predict the distribution behaviour of a nozzle. The spray pattern, in terms of flux density distribution, of a single-orifice nozzle was measured in an experiment. A CFD model, Fire Dynamic Simulator (FDS) beta version 6, was used for the numerical simulation. The data obtained from the experimental and numerical studies are compared.

Introduction

Water-mist fire suppression systems (WMFSS) represent a promising technology for a variety of applications within the fields of fire protection [1]. However, the performance of WMFSS greatly depends on the nature of a sprays produced by nozzles. The parameters that determine the behaviour of a spray are the water flow rate, spray angle, spray height, droplet size and droplet velocity. There have been numerous efforts to characterize the behaviour of sprays produced by nozzles and research is still on going to understand the influence of these parameters on spray dynamics [1, 2].

The dynamics of sprays can be investigated in two ways, i.e. by experiment or numerical simulation. Experimental investigation of behaviour of sprays produced by nozzles is very resource intensive. In comparison to this, computational modelling, with appropriate validation, is a tool to investigate the dynamics of sprays with various options of input parameters. There are numerous computational studies [3-9], which, mainly, focused on the interaction of water sprays with a fire plume, performance of sprinklers and extinction time of fires; but, only limited researches have been conducted on the characteristics of sprays produced by water-mist nozzles [1]. Without detailed knowledge of characteristics of sprays, typically quantified volume flux of water flow to the floor provides a little information [10]. There are various experimental [11-15] and numerical studies [10, 16-17] on the characterisation of spray patterns; however, most of them are related to sprays from conventional sprinklers. As water-mist droplet size is less than conventional sprinkler spray droplet, the dynamics of sprays of water-mist differ from that of conventional sprinklers. Hence, it is essential to quantify the characteristics of sprays produced by water-mist nozzles.

The objective of this study is to investigate the characteristics of a spray, in terms of flux density distribution, produced by a water-mist nozzle, both experimentally and numerically. An experiment was conducted using a single-orifice nozzle. A CFD simulation, using FDS6, was carried out with the same input parameters, used in the experiment.

Experimental Set-up

An experiment rig was constructed to measure the distribution of flux density of a spray. A single orifice nozzle of 1.524 mm diameter, discharging parallel to the nozzle axis, was used to produce the spray. A pump was installed with a capacity of 400 bar to supply water from a reservoir to the nozzle. A $2\times2\times0.1$ m water collection tray was placed beneath the nozzle. The tray was divided into 400 compartments, each with dimension of $10\times10\times10$ cm. The nozzle head was clamped 2.3 m above the floor. The experimental set-up and water collection tray of the test rig are shown in figures 1 and 2, respectively.

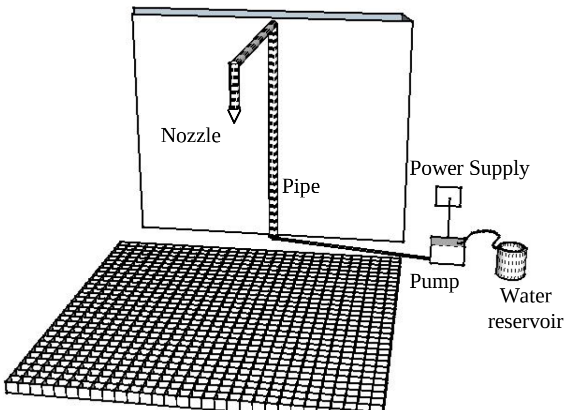

Figure 1. Schematic view of the experimental set-up.

Figure 2. Photograph of the water collection tray.

The pump was operated at a pressure which produced a flow rate of 1.7 L/min. The nozzle was allowed to operate until a stable spray was observed. The distributed water from the nozzle was collected on the tray for 180 seconds and the collected water in the box was measured in volume. The flux density distribution was calculated in $L/m^2/min$.

The spray angle of the nozzle was also measured in the experiment from the photographs of spray, which is illustrated in figure 3. The parameters of water flow rate, spray angle and spray height of the experiment were used in an FDS model.

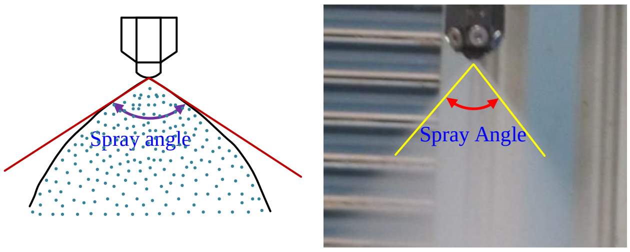

Figure 3. Measurement of spray angle, (a) Schematic view of measurement (b) Photograph of spray from which angle was measured.

Numerical Scheme

The CFD simulation was run using FDS6 (beta version). FDS is a CFD model, which has been developed by the Building and Fire Research Laboratory of the National Institute of Standards and Technology (NIST), USA. The finite difference technique was applied to solve the partial differential equation of conservation of mass, momentum and energy. In the following few sub-sections, the spray model used in FDS is described briefly. Details of these models are given in FDS technical reference guide [18].

Droplet distribution model

FDS takes a sample of spherical droplets to calculate the distribution pattern. The droplet size distribution is expressed in terms of its Cumulative Volume Fraction (CVF), which is represented by a combination of log-normal and Rosin-Rammler distributions [18].

$$F(d) = \begin{cases} (2\pi)^{-\frac{1}{2}} \int_0^{D_{CVF}} (\sigma D)^{-1} e^{\frac{[\ln(\frac{D}{D_m})]^2}{2\sigma^2}} dD & (D_{CVF} \leq D_m) \\ 1 - e^{-0.693(D_{CVF}/D_m)^{\gamma}} & (D_{CVF} > D_m) \end{cases} \quad (1)$$

where, D is the generic droplet diameter, D_m is the median droplet diameter. The median droplet diameter is a function of the sprinkler/nozzle orifice diameter, operating pressure, and geometry. γ and σ are empirical constants used for curve-fitting of distribution patterns.

Droplet transport model

In FDS, water droplet transport is modelled by Lagrangina approach. The velocity and position of droplets are calculated from the theory of conservation of momentum. The position and velocity of each droplet is calculated from the following equations.

$$\frac{d}{dt}(mv_p) = mg - \frac{1}{2}\rho C_d \pi r^2 (v_p - v_a)|v_d - v_a| \quad (2)$$

$$\frac{dx_p}{dt} = v_p \quad (3)$$

where, the drag coefficient, C_d, depends primarily on the Reynolds number based on the droplet-air relative velocity, which can be defined by,

$$C_d = \begin{cases} 24/Re & Re < 1 \\ 24(0.85 + 0.15Re^{0.687})/Re & 1 < Re < 1000 \\ 0.44 & Re > 1000 \end{cases} \quad (4)$$

Reynolds number of droplet is defined by

$$Re = \frac{\rho |v_d - v_a| \, 2r}{\mu(T)} \quad (5)$$

where $\mu(T)$ is the dynamic viscosity of air at temperature T.

Computational Model

Domain setup

A room of 2×2×2.4 m size was created as a computational domain. A grid sensitivity analysis was carried out with 10 cm, 5 cm and 2.5 cm cell sizes to select the appropriate grid size. The flux density distribution for these three cell sizes were calculated and compared with each other. The distribution of flux density for 5 cm and 2.5 cm were found to be almost identical. Therefore, a mesh of 5cm cell size was used to discretise the computational domain. The total number of cells in the domain was 76800. A water-mist nozzle was placed at a height of 2.3 m. Figure 4 illustrates the domain set-up of the numerical model. The green dots on the floor were the points where the water accumulation rate and the top one is the nozzle at 2.3 m height.

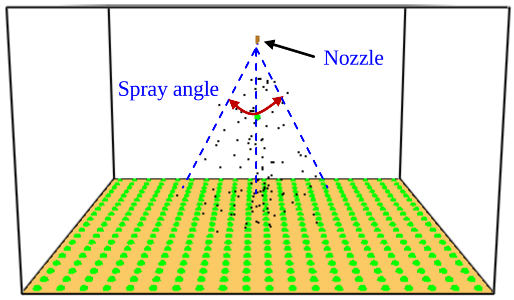

Figure 4. Computational domain set-up of the numerical model.

The floor was divided into 400 grids in XY plan with 10 cm grid size in each direction. The water volume accumulation rate were calculated in each grid of the floor. The grid system on the floor of the domain is illustrated in figure 5.

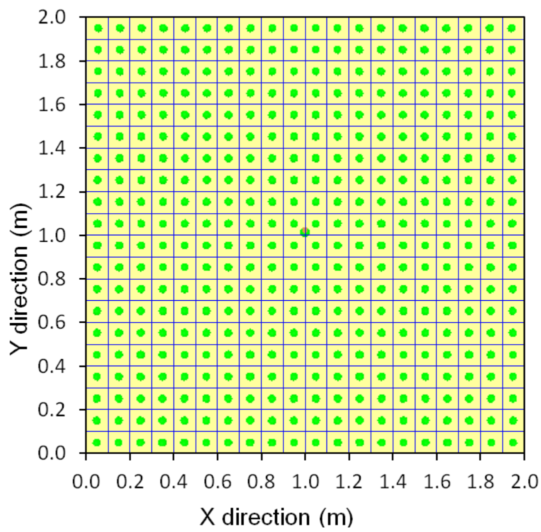

Figure 5. The grid system on the floor in the computational domain.

A Rosin-Rammler-lognormal distribution pattern was used for the distribution of drop size. The simulation was allowed to run for 65 seconds; the nozzle was activated at the beginning of the simulation and stopped at 60 seconds; the additional 5 seconds were to allow the water drops to fall down from the nozzle, based on preliminary calculation.

Input variables

Once the computational domain was set-up, the input parameters of the computational measurements were incorporated in the model. The input variables were the water flow rate, spray angle, spray height and droplet velocity. The volume accumulation rate on the floor was calculated from the simulation. The input parameters of the simulation are tabulated in table 1.

Flow rate	1.7 L/min
K-Factor	$0.073\ \mathrm{gpm/psi^{1/2}}$
Spray pattern type	Solid cone
Spray angle	65°
Spray height	2.3 m
Droplet velocity	$15.5\ \mathrm{m/s}$

Table 1. Input parameters for the numerical model.

Results and Discussion

In the experimental study, the flux density distribution of water spray was measured at a distance 2.3-m below of the nozzle. The input parameters similar to the experimental condition were used in the FDS model to mimic the experimental environment in the numerical model. These include water flow rate, droplet velocity, spray angle and spray height. Then, the flux density distribution of spray was calculated. The flux density distribution of experiment and simulation were calculated in $\mathrm{L/m^2/min}$ and the contour maps are illustrated in figures 6 and 7, respectively.

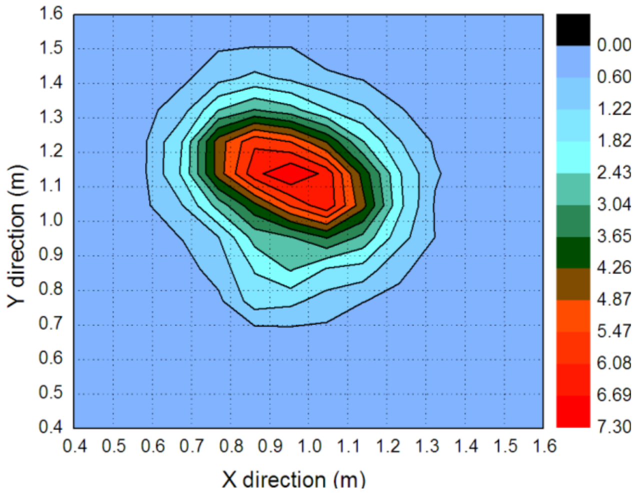

Figure 6. Contour maps of experimental values of water flux density distribution in $\mathrm{L/m^2/min}$ on the floor.

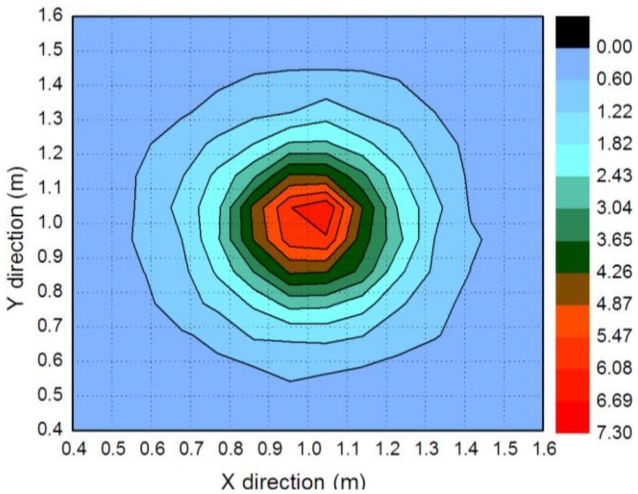

Figure 7. Contour maps of numerical values of water flux density distribution in $\mathrm{L/m^2/min}$ on the floor.

The contour maps are drawn from ordinate 0.4 to 1.6 in both X- and Y-axes (see figure 3), as 90% of water of spray was within this region of the water collection tray. The intensity of water distribution rate was highest at the centre of the contour maps, for both experiment and numerical model and it were decreased in the radial direction from the centre. The contour maps in figures 6 and 7 represent that there is discrepancy between the experimental and numerical contour maps. Uncertainty in the manual measurement of volume of water in each compartment can be responsible for the discrepancy in the experimental and numerical results, as there is likely to be some meniscus error in the manual measurement. Another thing is that the contour map of the experimental distribution is elliptical shape in X direction and it has moved 15 cm away from the centre of the tray to the positive Y direction. A possible reason is that the spray was drifted in this direction as a draft was generated in the vicinity of the spray. The reason for the generation of this draft is being investigated. In the continuation of this study, effects will be made to eliminate the draft or an additional parameter will be putted in the model as an input parameter to simulate the draft itself. Other than the elliptical shape of the experimental maps, the predicted distribution pattern of the numerical model is reasonably agreed with that of the experiment.

The distribution rates of water volume flux along the axes of the ellipse/circle are illustrated in figure 8. In the experimental study, the distribution patterns along the major and minor axes of the ellipse (see figures 6) are not identical, whereas the simulation results along the centreline (see figure 7) showed an almost identical distribution patterns. However, both experimental and numerical data shows a bell shaped distribution patterns.

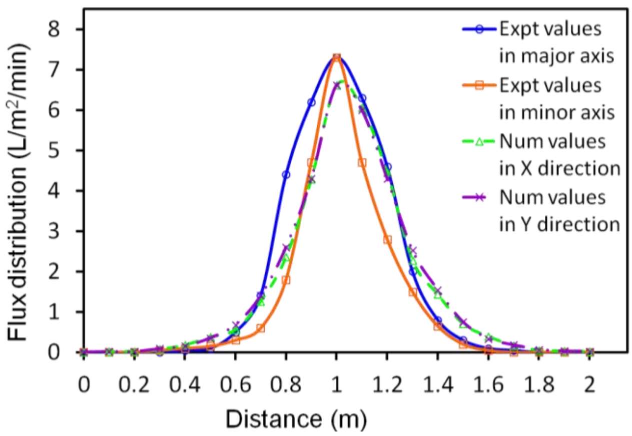

Figure 8. The distribution rates of spray along the centreline axes.

Conclusions

In this paper, the flux density distribution pattern of a single orifice water-mist nozzle is presented. From the analysis of contour maps of flux density distribution, it has been found that the CFD model has predicted the experimental values of distribution quite well. However, the shape of experimental map is elliptical, which is due to draft and efforts will be made to eliminate the draft or draft will be putted as an input parameter in the numerical model to mimic the experimental environmental in the model.

The distribution rates of water volume in the orthogonal directions are also presented. The patterns of distribution have been found to be almost identical along the centreline axes for the numerical model, whereas the experimental patterns have not been found to be identical in the orthogonal directions of the ellipse. A probable reason of this difference is that the spray was drifted in the experiment due to draft. However, the predicted patterns of the distribution rates are in reasonable agreement with the experimental values.

In the present study, the spray characteristics of a single orifice nozzle were studied. However, in future work, the spray characteristics of a multi-orifice water-mist nozzle will be studied. Furthermore, the effect of other influential parameters on behaviour of sprays, i.e. the effect of different flow rates, spray angles and spray heights will be investigated, for both single and multi-orifice nozzle.

Acknowledgement

The authors wish to acknowledge the technical and financial assistance provided by Defence Science Technology Organisation (DSTO), Australia.

References

[1] Santangelo, P.E., Characterization of high-pressure water-mist sprays: Experimental analysis of droplet size and dispersion, *Experimental Thermal and Fluid Science*, 2010, 34:1353–1366.

[2] Kim, S.C. & Ryou, H.S., An experimental and numerical study on fire suppression using a water-mist in an enclosure. *Building and Environment*, 2003; 38:1309-13016.

[3] Li, K.Y., Spearpoint, M.J., Ji, J., Huo, R., Li, Y.Z. & Hu, L.H., A mathematical model of the drag component of a sprinkler spray adjacent to horizontal smoke vents, *Journal of Fire Protection Engineering*, 2010; 20:2027-54.

[4] Li, K.Y., Hua, L.H., Huo, R., Li, Y.Z., Chen, Z.B., Li, S.C., Sun, X.Q., A mathematical model on interaction of smoke layer with sprinkler spray, *Fire Safety Journal*, 2009; 44:96– 105.

[5] Schwille, J.A. & Lueptow, R.M., A simplified model of the effect of a fire sprinkler spray on a buoyant fire plume. *Journal of Fire Protection Engineering*, 2006; 16:131-153.

[6] Marshall, A.W. & Marzo, M.Di., Modeling aspects of sprinkler spray dynamics in fires, *Process. Safety and Environmental Protection Part B,* 2004; 82(B2): 97–104.

[7] Nam, S., Numerical Simulation of the Penetration Capability of Sprinkler Sprays, *Fire Safety Journal*, 1999; 32(4):307-329.

[8] Nam, S., Development of a computational model simulating the interaction between a fire plume and a sprinkler spray, *Fire Safety Journal*, 1996; 26(1):1–33.

[9] Jackman, L.A., Sprinkler Spray Interaction with Fire Gases, *Ph.D. Thesis, South Bank University*, London, UK, 1992.

[10] Ren, N., Baum, H.R. & Marshall, A.W., A comprehensive methodology for characterizing sprinkler sprays. *Proceedings of the Combustion Institute,* 2011; 33: 2547–2554.

[11] Widmann, J.F., Phase Doppler interferometry measurements in water sprays produced by residential fire sprinklers, *Fire Safety Journal*, 2001; 36(6):545–567.

[12] Putorti, A.D., Belsinger, T.D., Twilley, W.H., Determination of water spray drop size and velocity from a low pressure, high momentum, water-mist nozzle. *Test report FR 4000, Building and Fire Research Laboratory,* National Institute of Standards and Technology (NIST), Gaithersburg, MD 20899, May 1995.

[13] Chan, T.S., Measurements of water density and drop size distributions of selected Esfr sprinklers, *Journal of Fire Protection Engineering*, 1994; 6(2):79–87.

[14] Chow, W.K. & Shek, L.C., Physical properties of a sprinkler water spray, *Fire and Materials*, 1993; 17:279-292.

[15] Lawson, J.R., Walton, W.D. & Evans, D.D., Measurement of droplet size in sprinkler sprays, *Report No. NBSIR 88-3715, Center for Fire Research,* National Engineering Laboratory, National Bureau of Standards, Gaithersburg, MD 20899, 1988.

[16] McGrattan, K.B., NIST sponsored research in sprinkler performance modelling, Research and Practice: Bridging the Gap, Fire Suppression and Detection Research Application Symposium. *Proceedings. Fire Protection Research Foundation*, Orlando, FL, February 7-9, 2001, 596-603.

[17] Chow, W.K. & Yin, R., Water spray pattern discharged from high headroom atrium sprinkler. *Journal of Architectural Engineering*, December 1999, 133-140.

[18] McGrattan, K.B., Hostikka, S., Floyd, J., Baum, H., Rehm, R., Mell, W. & Macdermott, W., Fire Dynamics Simulator (Version 5), Technical Reference Guide, Volume 1: Mathematical Model, *NIST Special Publication 1018-5*, National Institute of Standards and Technology (NIST), U.S. Department of Commerce, Gaithersburg, Maryland, USA, 2010.