A novel approach for simulating droplet transport for watermist applications

Vaidyanathan Sankaran, Ragini Acharya & Marios Soteriou
United Technologies Research Center
411 Silver Lane, East Hartford, CT 06108

1. Introduction

In recent years, use of watermist systems for fire suppression is gaining popularity in a wide variety of applications ranging from ships to commercial building protection. Watermist systems control and/or suppress a fire by three main mechanisms (1) cooling (2) inerting and (3) blocking of radiation. Unfortunately there are no generic principles to guide the design of an injector (nozzle) that can yield optimal drop size distribution to achieve higher penetration and three-dimensional dispersion simultaneously for the above mentioned mechanisms to be effective. Therefore water mist systems are evaluated using a “performance-based” approach in which fire tests are designed to assess the performance limits of the system. These tests are usually expensive and are not generic. Therefore there is a strong need for predictive fire suppression modeling tools which can be used to assess the performance of watermist systems. As a first step, for a modeling tool to be useful for design of watermist injectors, the model must be able to predict droplet distribution reasonably well to yield accurate patternation data on the floor, droplet flux, etc. Computational Fluid Dynamics (CFD) based modeling tools have been in use to model spray systems for various applications (El Banhawy et al. (1980), Faeth, (1987)) and UTRC has a long history in developing spray models with varying levels of fidelity for aero-engine combustors (Madabushi, 2003, Li et al (2010))). However, to be able to predict the droplet distributions accurately, one must resolve the complex droplet formation processes (atomization) accurately and in spite of the recent advances in computational simulations of atomization processes (Sussman et al (2007), Li et al (2010)) extension to practical systems remains difficult.

2. Review of prior work

Current multi-phase models make no attempt to characterize the flow-field within the injector to predict spray formation processes. Instead, most of the computational models (El Banhawy et al. (1980), Faeth, (1987), Madabushi, 2003), neglect the atomization processes altogether and model the liquid jet issuing from an injector as a collection of discrete droplets that satisfy integral conservation like mass and momentum flux. This computational approach has been used in several multi-phase flow applications with reasonable success. However, the accuracy of the solution strongly depends on the initial and boundary conditions for the droplets, such as droplet size and velocity distribution functions - which are difficult to obtain very close to the injector by experimental means. As a result, most of the CFD models use one of the two following empirical approaches: (1) use/extrapolate the data measured at a downstream location for the inlet boundary conditions at the nozzle exit or (2) assume a droplet size distribution at the injector exit and iterate on the parameters to match/fit the measured data at the downstream location. For the droplet velocity distribution, it is usually assumed that all droplets emerge from the injector at the same average velocity even though experimental results (Presser et al. (1993)) show a broad distribution. These empirical methods are usually applicable only for a limited range of conditions and hence requires tuning for every nozzle and operating conditions to match the observed data. This *adhoc* tuning of parameters is non-trivial and time consuming and may not necessarily satisfy the constraints imposed by the conservation laws of mass, momentum and energy.

The specific objective of this work is to devise a methodology free of empiricisms, applicable to a wide range of nozzles/conditions and one which satisfies the conservation laws of mass, momentum and energy.

3. Technical Approach

The crux of this approach is to use detailed droplet data from the a *high fidelity experimental measurements* that was collected sufficiently close to the injector and still far enough that one can ignore the effects of liquid jet break-up, atomization physics and dense spray evolution. The novelty of the current approach lies in the way the experimental data is coupled to the CFD model. The three key elements of our approach are highlighted here

(1) ***Mass & Momentum Scaling:*** Due to the inherent limitations in the measurement technique (and due to other errors), not all the droplets that pass through the measurement probe volume can be measured. Therefore, the measured data may not contain information about all the droplets of the spray and there could be a significant loss of mass and momentum at the measurement location compared to the initial conditions at the orifice. Therefore, we corrected the lost/missing information by scaling the droplet size and velocity distribution, in order to satisfy the conservation of mass and momentum.

(2) ***Entrainment Boundary Conditions****:* It is not enough to specify the droplet conditions correctly, without accounting for the gas entrainment, as this would under-predict the droplet penetration. But unfortunately, entrained gas velocity data at the droplet location is not typically available because of the difficulty in measuring gas and droplet velocity simultaneously. Therefore, we have devised a methodology to account for the gas entrainment due to droplet motion and included in our model.

(3) ***Internal Computational Boundary:*** As explained in the previous section, it is erroneous and time consuming to use the data measured at a downstream location at the nozzle exit plane location in the CFD model. To avoid this, we specify the droplet size and velocity distribution at the same location where it was measured at every time-step of the calculation. This precludes the need to iterate on the initial conditions to match the measured data and make the calculation more predictive and cheaper.

4. Results & Discussion

We applied this approach to a single-orifice injector that was installed to spray vertically downward (axial direction) in a rectangular room. Droplet data was measured using PDI technique at two axial planes and at each axial plane, droplet size and velocity distributions were measured at several radial locations. Large Eddy Simulation (LES) was used to model this problem and the entire room that included the injector exit was meshed. The Fire Dynamics Simulator (FDS) code developed by NIST was used to simulate this problem. An internal computational boundary was defined in the CFD at a location where the first measurement plane was located in the experiments. Note: This plane is a very short distance away from the injector exit as compared to the room size. At this internal computational boundary, measured droplet size and velocity distributions are specified (after correcting for the mass/momentum loss) following the approach explained in the previous section. Figure 1(a) and 1(b) shows the droplet distribution and entrained gas velocity, respectively at a certain instant which exhibits good qualitative agreement with the experiments. Figure 1(c) compares the axial droplet velocity data measured at a second measurement plane with the model prediction. A very good agreement is seen between the model and the experimental data. The same model was also implemented in a commercial software Fluent and a similar quantitative agreement (results not shown here) with the experiments was obtained.

Upon successful validation of the approach in a single-orifice injector, the same approach was extended to a multi-orifice injector. A typical water mist injector is shown in fig. 2 (Source: www.sfpe.org) and it usually consists of a central orifice surrounded by several "satellite" orifices positioned on the periphery of the spray head at different angles from the vertical direction. In order to model this multi-orifice injector, the droplet data from the single orifice injector was rotated and mapped to each and every satellite orifice on the periphery of a typical water-mist nozzle. LES was used to model the droplet transport from the water-mist injector in a full room.

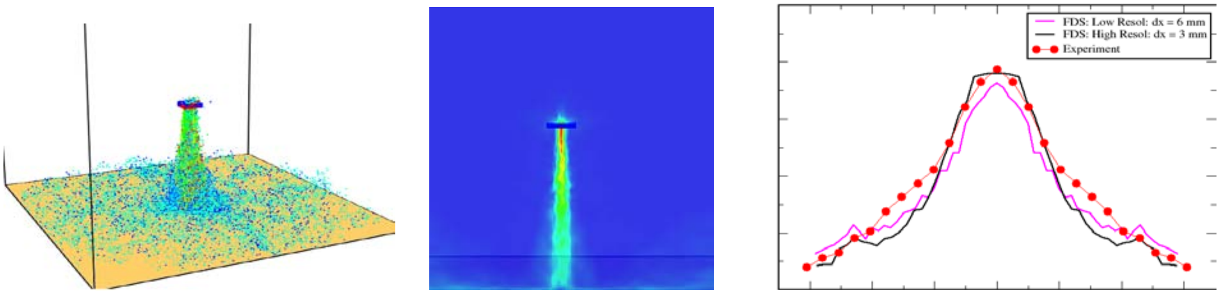

Figure 1: (a) snap-shot of droplet distribution (b) snap-shot of entrained gas velocity (c) droplet axial velocity at the second measurement plane

Figures 2(b) and 2(c) shows an instantaneous snapshot of the droplet distribution in the room and note that the plane from where the droplets originate in this calculation is not from the orifice exit but instead from the first measurement plane in the experiments. Comparison of the droplet distribution pattern on the floor with experiments (not shown here) showed reasonable agreements. Size and location of the foot-print of the spray jets emanating from different orifices of the injector compared well with the measurement.

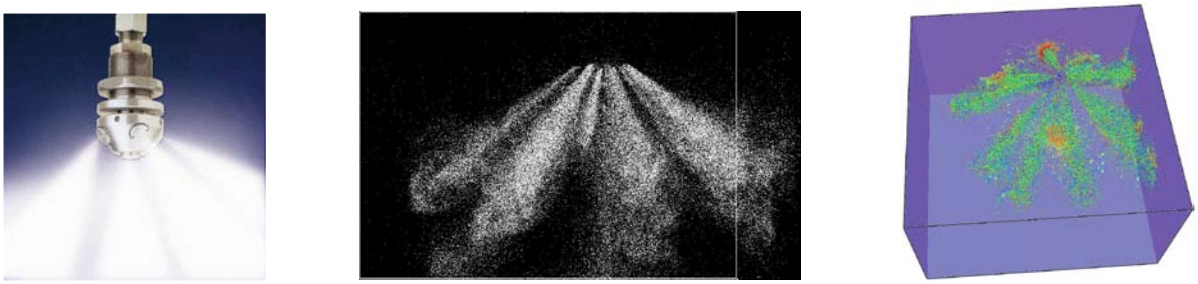

Figure 2: (a) a typical water-mist injector (b) snap-shot of droplet distribution (c) perspective view of the droplet distribution (colored by size) in the room

Further work is underway to compare other metrics such as droplet size and velocity distribution at different locations in the room. Future work may include other physics of fire suppression, such as buoyancy effects on droplet distribution and interaction of droplets with burning surfaces.

5. References

(1) Sussman, M., Smith, K. M., Hussaini, M.Y., Ohta, M., and Zhi-Wei, R., *Journal of computational physics*, Vol. 221, pp. 469-505, 2007
(2) Li, X., Arienti, M., Soteriou, M., AIAA-2010, *48th AIAA Aerospace Sciences Meeting*, Orlando, FL, 2010.
(3) El Banhawy, Y., and Whitelaw, J., *AIAA Journal*, Vol. 18, pp. 1503, 1980
(4) Faeth GM., *Progress in Energy and Combust Sciences* 1987; Vol.13, pp: 293, 1987
(5) Madabushi, R.K., *Atomization and Sprays*, Vol. 13, pp. 413-424, 2003
(6) Presser, C., Gupta, A. K., Semerjian, H. G., *Combustion and Flame*, Vol. 92, pp.25, 1993