STUDY OF WATER MIST SUPPRESSION OF ELECTRICAL FIRES FOR SPACECRAFT APPLICATIONS: NORMAL-GRAVITY RESULTS

Angel Abbud-Madrid, Sonny J. Lewis, James D. Watson, and J. Thomas McKinnon
Center for Commercial Applications of Combustion in Space
Colorado School of Mines
Golden, CO 80401
Ph: (303) 384-2300, E-mail: aabbudma@mines.edu

Jean-Pierre Delplanque
Mechanical and Aeronautical Engineering
University of California
Davis, CA 95616-5294
Ph: (530) 754-6950, Email: delplanque@ucdavis.edu

ABSTRACT

A preliminary investigation on the effectiveness of water mist as a suppressant in electrical fires under normal-gravity conditions for spacecraft applications is presented. Water-mist suppression experiments of a fire involving an overheated wire are conducted inside a container similar to the Space Shuttle mid-deck locker. Direct and indirect water-mist injection is used with various droplet-size distributions and flow rates. Water mist quickly extinguishes a fire that is directly impacted by the droplets, while much longer spraying times and larger amounts of water are required to suppress fires burning behind a baffle. Smaller droplet size distributions appear to be the most effective. A numerical model enables the simulation of a polydispersed spray, while still providing enough droplet scale resolution for the high-gradient fire suppression scenarios. The preliminary numerical results accurately predict droplet penetration, evaporation, and dispersion into the container as observed in the normal-gravity tests. These qualitative comparisons contribute to the on-going validation process of the model.

INTRODUCTION

The renewed emphasis on the human exploration of space is focusing on the development of new spacecraft like the Crew Exploration Vehicle (CEV) and on future planetary habitats for the long-term settlement of the Moon and Mars. The development of these new programs has consequently prompted a reevaluation of current fire suppression systems on spacecraft and it has motivated a feasibility study for possible replacement technologies. The challenges to the designer of a new fire suppression system for space applications are many and sometimes unique to the type of environment encountered outside the Earth's atmosphere and in other planets. The use of a light, non-toxic, and efficient suppressant capable of rapidly extinguishing a fire in a confined space with minimum generation of toxic byproducts and with fast and easy cleaning and recovery procedures are among these challenges. For long duration missions, the ability to refill the extinguisher with an agent easily available in the spacecraft is also of primary concern. In selecting such a suppressant agent it is necessary not only to look at its extinction efficiency as compared to other options, but it is also important to study the dispersion properties of the agent in partial gravity environments and in the presence of complicated geometries with a variety of obstacles, ventilation sources, and fire scenarios.

In a preliminary evaluation of the various suppressant agents available, it appears that water mist may be a good candidate to address most of the above challenges. On a per unit-mass basis, water is as effective as Halon 1301, the agent currently used in the Space Shuttle, while water is more effective than carbon dioxide (CO_2), the agent onboard the International Space Station. Water is also non-toxic, non-corrosive, readily available in spacecraft for multiple uses, and water in the form of ultra-fine mist may act as a total flooding agent in reduced gravity. In addition, advantage may be taken of the rapid evaporation of ultra-fine mist for its use in fighting electrical fires. Finally, agent cleanup operations may be achieved with dehumidifiers in the ventilation system. Consequently, the suppression properties of water mist are currently being investigated in the search for new fire extinguisher systems for the next generation of spacecraft.

As a result of the motivating factors mentioned above, a comprehensive study of the fire suppression properties of water mist in spacecraft and extraterrestrial habitats is being conducted at the Center for Commercial Applications of Combustion in Space at the Colorado School of Mines. The purpose of this project is to investigate the effectiveness of water mist in single or mixed-agent configurations on different fire scenarios, geometries, and low-gravity conditions evaluated numerically and experimentally and compared to other fire-fighting agents currently used in spacecraft fire-safety systems. The modeling effort consists of developing detailed sub-models of the fire source, the suppression agent generation and distribution, and the radiative shielding of the suppression agent. These sub-models will then be integrated into a high-fidelity, fire-suppression model. Finally, a reduced order model will be developed to minimize the computational requirements, yet retain the simulation capabilities of the original formulation. This paper captures the experimental and numerical modeling work done to date, which has focused on the preliminary evaluation of the effectiveness of water mist as a suppressant agent in electrical fires under normal-gravity conditions. The experimental work concentrates on evaluating the suppression effects of droplet size distribution and the behavior of water mist in a constrained geometry. In a parallel effort, the numerical work focuses on the simulation of the generation, evaporation, and distribution of the mist as it moves through the container and as it interacts with the fire source.

EXPERIMENTAL SETUP

The first set of experiments conducted under this program has been performed in a 44-cm wide, 25-cm high, and 51-cm deep container with similar characteristics to the Space Shuttle mid-deck locker, as described in a previous publication [1] and as shown in Fig. 1. Since an overheated-wire failure has been identified as one of the most probable fire scenarios to occur in a spacecraft, suppression experiments are conducted with a 15-cm long, polyethylene-insulated #20 wire with a high current flowing through it. Burning behavior is observed and flame-spread and heat-release rates under a downward propagation configuration are measured. Although these tests are conducted in normal gravity with a buoyancy dominated flow field, these downwardly propagating flames exhibit a well-behaved flame front reminiscing of propagation under low-gravity conditions. In contrast, flames propagating in the horizontal and upward direction are plagued with instabilities and turbulence generated by buoyancy. A measure of the time from ignition to extinction of the flame, the mass of insulation burned, and the heat of combustion of polyethylene gives an average fire size of 72 W. The electrically heated wire raises the surface temperature of the wire insulation to over 100C with a current of 35 amps without leading to ignition. Raising the current level above 35 amps only causes the wire to distort and melt away the insulation without a transition to flaming. Thus, an external ignition source is needed to initiate a flame that can only be sustained by constantly heating the wire with electrical current.

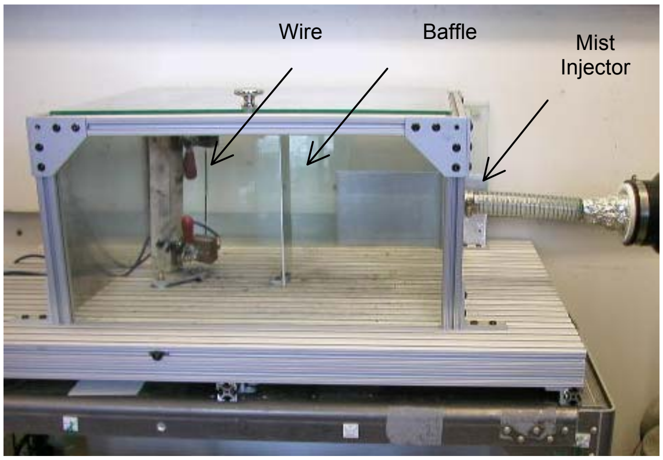
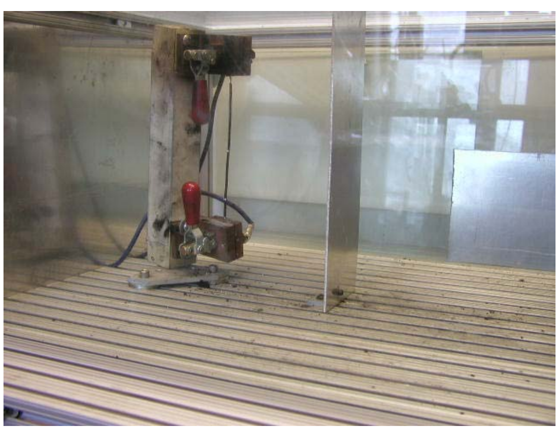
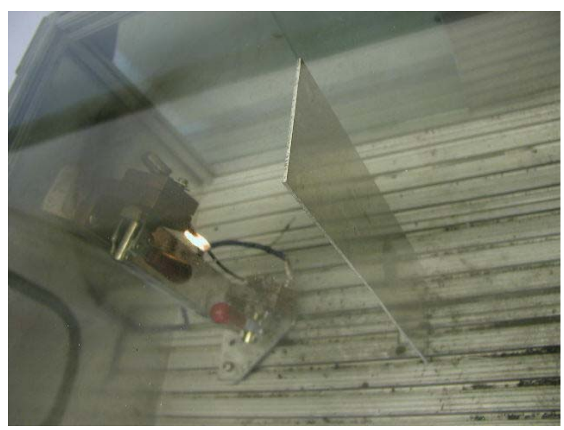


Figure 1. Experimental apparatus for electrical fire suppression tests based on the Space Shuttle mid-deck locker.



(a)



(b)

Figure 2. (a) Side view of the experimental setup showing the vertical test wire and the baffle in front of it, and (b) overhead view at the start of a test with a flame at the top of the wire.

In a typical test, the sample wire is held vertically between two large copper clamps. A current of 35 amps is applied and the wire is allowed to heat for 30 seconds before it is ignited near the top clamp with a propane lighter. After propagating for 2.5 cm, the burning time is measured for the next 5 cm to calculate the flame speed. The average downward flame speed is 0.06 cm/sec. Images of the experimental setup and the burning wire are shown in Fig. 2.

For the suppression tests, direct and indirect water-mist injection methods with a high-momentum jet are used. As shown in Fig. 1, the wire is located 40 cm from the water mist injector and for the indirect-injection tests a 13-cm wide, 25-cm high baffle is placed at 26 cm from the nozzle, in front of the burning wire. This last configuration is used to provide an extremely difficult path for the water mist to reach the burning wire. Interestingly, varying the width of the baffle from 5 to 13 cm had only a minor effect on the suppression efficacy of the high-momentum jet. Different droplet-size distributions and flow rates are possible by varying the water pressure on a water-mist injector with a 0.2-mm diameter orifice. Droplet size distributions with a Sauter mean diameter (SMD) of 40 to 27 μm are achieved with pressures varying from 100 to 1000 psi, respectively.

NUMERICAL MODEL

The mist dispersion and evaporation model used in this paper has been described previously [1] and is only outlined here for completeness. The mist is discretized using a Monte Carlo approach [2] that requires as inputs four parameters: diameter, speed and two angles of injection. The method assumes a log-normal drop size distribution, Gaussian droplet speed distribution, and uniformly distributed angles of injection. The droplets have the same temperature at injection (300K). The history of each representative droplet thus defined is then calculated as described below. Finally, overall mist behavior is reconstructed by integrating droplet histories in the Eulerian frame. This approach is somewhat similar to that proposed by Schmehl and co-workers [3]. More details regarding this approach are provided in [2].

Evaluation of the mist behavior requires a thorough understanding of in-flight droplet motion. The droplet-gas relative velocity significantly influences the vaporization rate of the droplet in-flight, as well as its trajectory and, therefore, mist penetration. To represent this phenomenon, a simplified version of the particle equation of motion is employed to track the droplets in a Lagrangian manner [4],

$$m_d \frac{d\vec{V}_d}{dt} = F_D + F_T \tag{1}$$

where m_d is the droplet mass, V_d is the droplet velocity, F_D is the drag force, and F_T is the thermophoretic force based on the temperature gradient in the continuous phase.

Depending upon the resolution required by the configuration considered, droplet thermal energy conservation may be evaluated using a lumped parameter approach or using the spherically symmetric transient conduction equation,

$$\frac{\partial T}{\partial t} = \alpha_{\text{eff}} \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 \frac{\partial T}{\partial r} \right) \tag{2}$$

where T is the droplet temperature, α_{eff} is the effective diffusivity, a symmetry condition is enforced at the droplet center, and convective heat transfer is applied at the droplet surface. While it is more computationally intensive than the lumped parameter approach, the resolution of the temperature distribution in the droplet more accurately portrays the evaporation process, especially in situations where the droplet is exposed to large temperature gradients, as is the case here.

As the droplet evaporates, the interface recedes and latent heat is absorbed [5]. The balance between the heat conducted in the droplet, the heat convected from the carrier gas and the enthalpy of vaporization provides the boundary condition at the droplet surface. The evaporation of the droplet is evaluated using Abramzon and Sirignano's extended film model [6],

$$\dot{m} = 2\pi \widehat{\rho_g \mathcal{D}_g} R \, \mathrm{Sh}^{\star} \; \ln(1 + B_M) \tag{3}$$

where ρ_g is the droplet density, D_g is the diffusion coefficient of the continuous phase, B_M is the transfer number, and Sh^* is the modified Sherwood number which, in addition to the convective effects, accounts for the effect of Stefan flow on the mass transfer. The droplet surface temperature is an eigenvalue of the problem obtained from the film model and it is used to calculate the partial pressure of water vapor at the droplet surface using the Clausius-Clapeyron equation [7]. The droplet surface regression rate is then given by:

$$\frac{dR}{dT} = \frac{-\dot{m}}{4/3\pi \rho_\ell R^2} \tag{4}$$

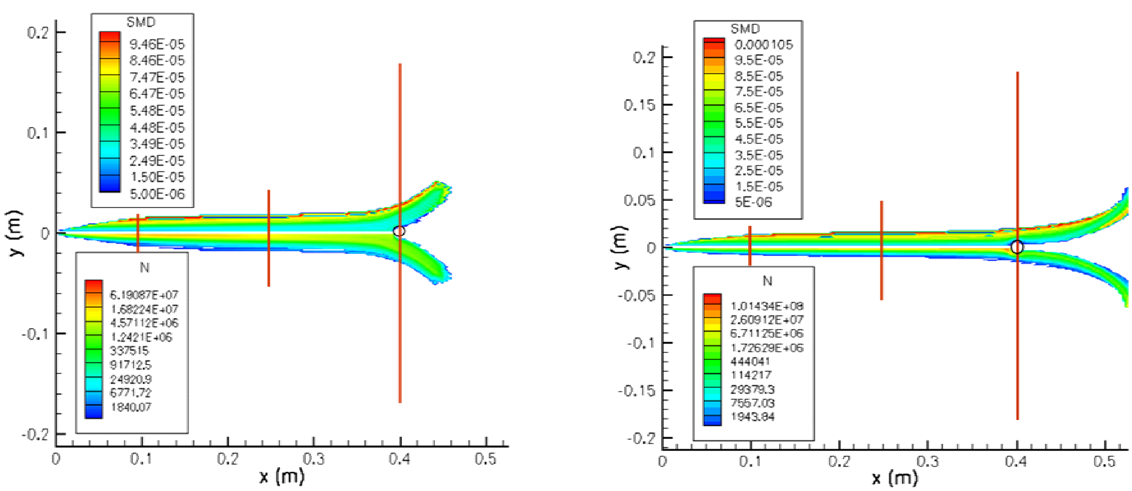
The local gas conditions needed at each location to solve for the droplet equations are obtained by interpolation in a pre-computed background flowfield using the CFD-ACE software code developed by the CFD Research Corporation.

RESULTS AND DISCUSSION

The preliminary tests show that water mist quickly extinguishes a fire that is directly impacted by the droplets, while much longer spraying times and larger amounts of water are required to have an effect on the fires burning behind a baffle. For a water pressure of 300 psi producing a spray with a Sauter mean diameter of 35 µm and a droplet velocity of 5 m/sec, the average extinguishment times for the direct and indirect injection cases were 10 seconds and 95 seconds, respectively. Similarly, the average amount of water used for extinction was 10 ml and 95 ml for the respective cases. For the latter, a gravimetric measurement of the water reaching the wire showed only a 1.5% of the total amount of water injected during the test, indicating that most of the droplets are captured by the baffle and the walls of the container. For larger droplet sizes at the lowest water pressures with the baffle present, the flames were only slightly slowed down, but never extinguished. While the direct injection method is much more effective in extinguishing a flame than the indirect injection method, both cases exhibit similar extinction behavior where the flame shows a brief oscillation between small and large size flames until extinction occurs. This type of pulsating extinction phenomena has been observed before in suppression experiments with water mist [8].

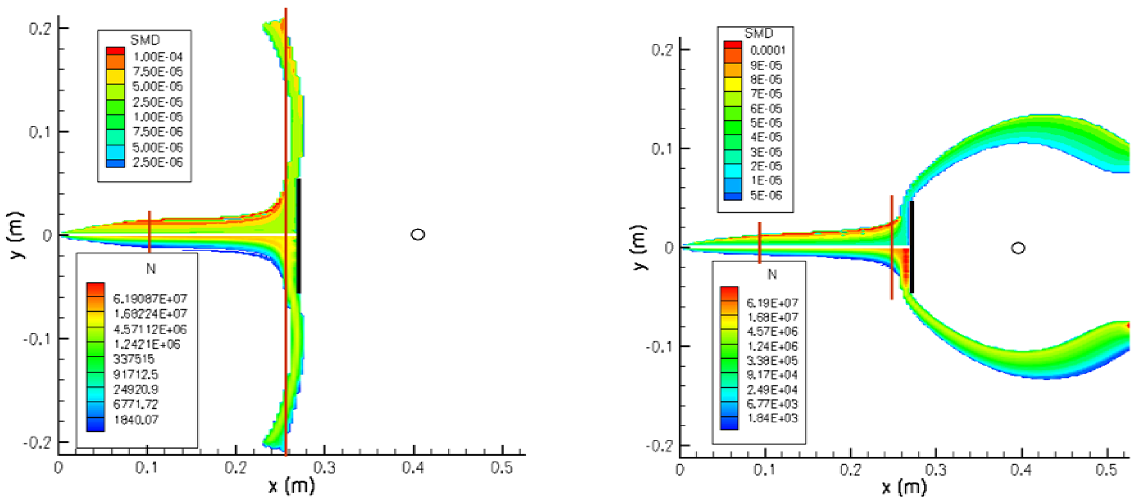
Using the numerical model described in the previous section and using the same mist characteristics as in the experimental tests, a limited parametric study of fire suppression by water mist was conducted. The numerical results show the dispersion and evaporation of droplets as they move towards the fire source in the container used in the normal-gravity tests. The numerical simulations occur in a 2-D configuration coinciding with the experimental locker setup to predict the fire suppression ability of the mist. The four relevant configurations are constructed by varying two parameters: the presence of an obstacle, and a coflow in the locker. The four spray development plots in Figs. 3 and 4 show the resulting spray conditions present in each of the four cases. The presence of the coflow greatly influences the temperature field seen by the mist. In the two cases with the coflow the locker temperature is lower, because the residence time in the chamber is affected by the coflow. The presence of the baffle also creates two circulation zones, divided by the baffle. These zones greatly affect the ability of the droplets to interact with the burning wire and are highly dependent on the position of the two outlets on the wall behind the burning wire. The circulation zones also will affect the temperature field seen by the droplets.

The fire suppression ability is exhibited by the Sauter mean diameter (SMD) and number density (N) profiles produced by the numerical simulations. The two cases evolving in the field without the obstacle, exhibit similar flow patterns, and both have direct spray interaction, where the high latent heat of vaporization of water can be utilized. The direct interaction scenario is ideal for fire suppression. The coflow also influences the suppression ability. In the specific burning wire case, the coflow assists in focusing the spray at the target as seen in Fig. 3b. This is evident in the comparison of the two cases. The number density in the coflow assisted case is two orders of magnitude higher, thereby increasing the enthalpy removal from the flame.



(a) (b)

Figure 3. Predicted water-mist Sauter mean diameter (SMD) and number density (N) profiles with (a) no coflow and (b) a 5 m/s coflow.



(a) (b)

Figure 4. Predicted water-mist Sauter mean diameter (SMD) and number density (N) profiles evolving around an obstacle with (a) no coflow and (b) a 5 m/s coflow.

The two direct spray scenarios are in stark contrast to the scenarios involving the baffles. In the latter, the obstacle provides a limitation to the effectiveness of the high-momentum mist injectors used. Figure 4 shows how the baffles serve to deflect the flow, causing the predicted Sauter mean diameter maps. The predicted behavior shows no direct interaction between the mist and the burning wire. In the case without the coflow, the droplets do not have significant inertia to move around the obstacle and instead a large recirculation region is generated, where the mist will eventually impinge on the chamber walls. In the case with the coflow however, the droplets get diverted and move around the obstacle. The reattachment length for the spray occurs past the chamber wall, making it impossible to have direct interaction with the wire. The experimental results show that the wire will interact with only about 1.5% of the mist by mass. The discretization process associated with the numerical model is not accurate to that low percentage level. Increasing the number of representative droplets would increase the model accuracy and might allow the description of this interaction. In the current scenarios, the lack of interaction would suggest, as confirmed by experiments, that fire suppression effects would be minimal, and they would mainly be due to mechanisms other than enthalpy extraction.

The radial SMD profiles elucidate the interaction conditions. Figure 5a shows the difference in the radial spread of the mist, at 10 cm away from the injector, in which the 5 m/s coflow concentrates the mist flow into the center, where the spray/wire interaction will be increased, effectively enhancing the suppression effect. Fig. 5b shows the divergence of the mist around the obstacles at a point 25 cm away from the injector and the eventual recirculation region of the mist without the coflow. The direct interaction seen in Fig. 5c at the wire location, 40 cm away from the injector, predicts a larger SMD in the case with the coflow. The number density plots in Fig. 3 would suggest that the total mass flow rate required for suppression would be less than the situation without the coflow.

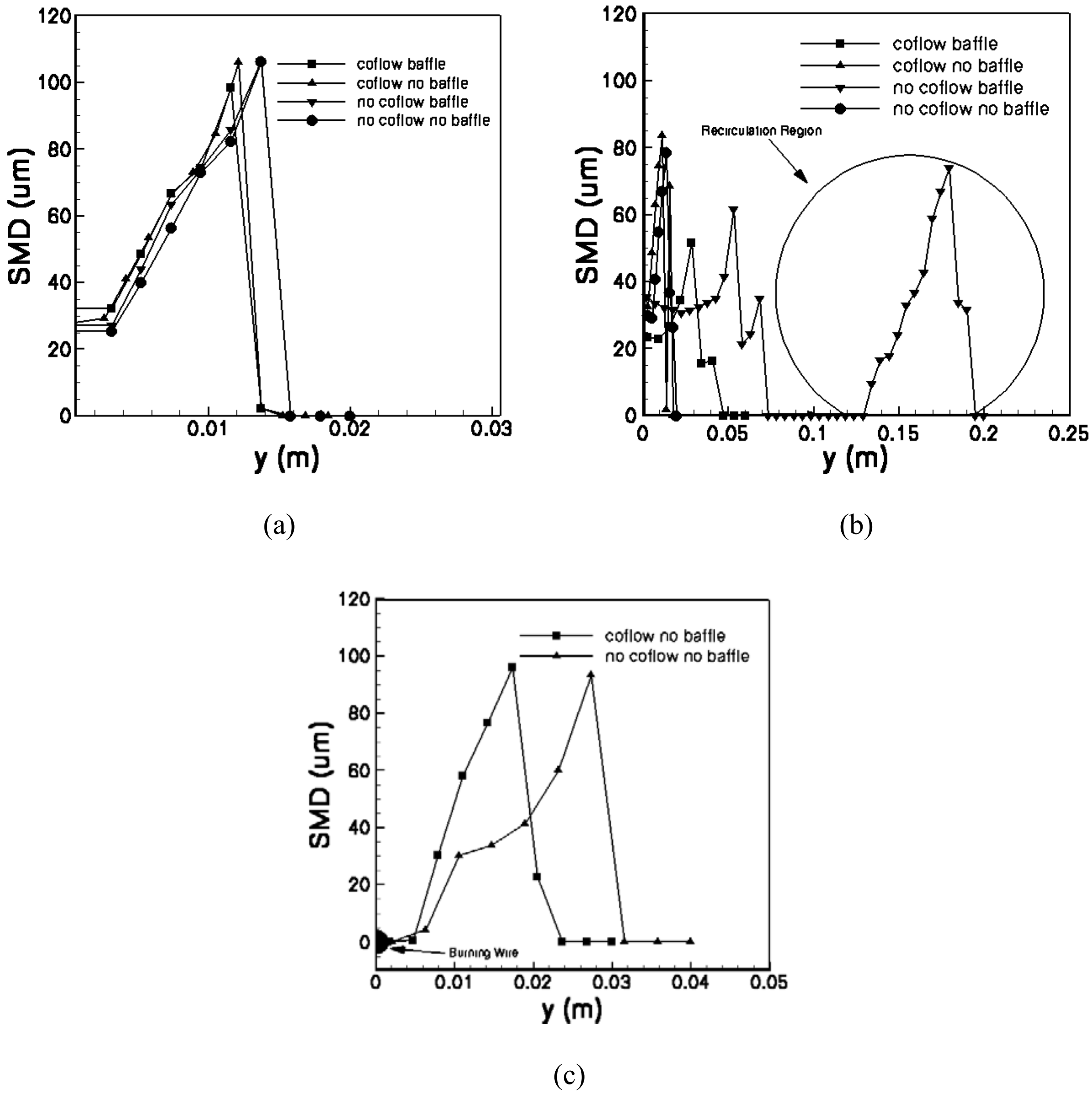


Figure 5. Radial SMD profiles at various axial locations: (a) 10 cm, (b) 25 cm, and (c) 40 cm, with the latter showing the interaction of the mist with the burning wire.

Based on the results obtained with the high-momentum, large-droplet-size jets, we have recently started a series of tests with low-momentum, ultra-fine mist for comparison purposes. For these tests, the high-pressure nozzles were replaced by an ultrasonic mist generator developed by NanoMist Systems, which generates a droplet size distribution with a SMD lower than 10 μm at flow speeds in the vicinity of 0.5 m/s. From the few cases run to date, it is clear to see that ultra-fine mist can easily go around the baffle and rapidly flood the entire container in a gas-like manner. For the same experimental conditions described in previous sections for the indirect-injection case with a baffle, the flame propagating down the insulated wire was extinguished by the ultra-fine mist with an average of 4 ml of water in approximately 10 seconds, an order of magnitude lower, in both extinction time and water amount, than the high-momentum, large-droplet-size spray jet.

CONCLUSIONS

A preliminary experimental and numerical investigation on the effectiveness of water mist as a suppressant in electrical fires under normal-gravity conditions for spacecraft applications has been conducted. Direct and indirect water-mist injection is used with various droplet-size distributions and flow rates to extinguish a burning wire inside a container similar to the Space Shuttle mid-deck locker. Water mist quickly extinguishes a fire that is directly impacted by the droplets. However, much longer spraying times and larger amounts of water are required to suppress fires burning behind a baffle. Extinction times and water amounts are reduced as the droplet size distribution is lowered. A numerical model simulating the generation, evaporation, and distribution of a polydispersed spray accurately captures the phenomena observed experimentally. This experimental-numerical comparison has provided valuable information in the on-going validation of the mist dispersion sub-model, which is part of a three sub-model suite (along with the fire and radiation sub-models) required for the final development of a reduced order model to describe the complete fire suppression process.

As a result of the ineffectiveness of high-momentum, large-droplet-size spray jets to maneuver around obstacles and extinguish obstructed fires, the research effort is now shifting to the study of low-momentum, ultra-fine mist as a technique to produce total flooding of the confined space and effective suppression even in complicated geometries with fires hidden by multiple obstacles. The preliminary results obtained on the suppression of a burning wire with ultra-fine mist show an order of magnitude reduction in the time and the amount of water needed to extinguish the fire. These promising results have prompted an evaluation of ultra-fine water mist as a potential agent for the suppression of the fire types most likely to be encountered in spacecraft applications. An experimental and numerical feasibility study of the fire suppression properties of ultra-fine water mist will constitute the next phase of the project.

ACKNOWLEDGMENTS

This work is supported by the National Aeronautics and Space Administration, under Grant NNC04AA13A. The authors wish to acknowledge the invaluable help of Dr. Suleyman Gokoglu, the project monitor from NASA Glenn Research Center, and the technical help of the complete team at NanoMist Systems who provided an ultra-fine mist unit for this study.

REFERENCES

1. Delplanque, J. P., Abbud-Madrid, A., McKinnon, J. T., Lewis, S. J., and Watson, J. D., "Feasibility Study of Water Mist for Spacecraft Fire Suppression," *Proceedings of the Halon Options Technical Working Conference (HOTWC-04)*, The University of New Mexico, Albuquerque, NM, May 2004.

2. Johnson, S. B., *Multi-Scale Modeling of Spray Processes,* Ph.D. Thesis, Colorado School of Mines, 2004.

3. Schmehl, R., Maier, G., and Wittig, S., "CFD Analysis of Fuel Atomization, Secondary Droplet Breakup, and Spray Dispersion in the Premix Duct of a LPP Combustor," *Eighth International Conference on Liquid Atomization and Spray Systems*, July 2000.

4. Delplanque, J. P. and Rangel, R. H., "Droplet-Stream Combustion in the Steady Boundary Layer near a Wall," *Combustion Science and Technology*, **78**, pp. 97–115, 1991.

5. Sirignano, W. A., *Fluid Dynamics and Transport of Droplets and Sprays*, Cambridge University Press, 1999.

6. Abramzon, B. and Sirignano, W. A., "Droplet Vaporization Model for Spray Combustion Calculations," *International Journal of Heat and Mass Transfer*, **32**, pp. 1605–1618, 1989.

7. Reid, R. C., Prausnitz, J. M., and Poling, B. E., *The properties of gases and liquids*, McGraw-Hill, New York, N. Y., 4th Ed., 1987.

8. Abbud-Madrid, A., McKinnon, J. T., Amon, F., and Gokoglu, S., "Suppression of Premixed Flames by Water Mist in Microgravity: Findings from the *MIST* Experiment on STS-107," *Proceedings of the Halon Options Technical Working Conference (HOTWC-04)*, The University of New Mexico, Albuquerque, NM, May 2004.