National Research Council Canada
Conseil national de recherches Canada

NRC · CNRC

REVIEW OF THREE DIMENSIONAL WATER FOG TECHNIQUES FOR FIREFIGHTING

Research report: RR124

Date of Issue: December 2002

Authors: Z. Liu, A. Kashef, G.D. Lougheed and N. Benichou

Published by
Institute for Research in Construction
National Research Council Canada
Ottawa, Canada
K1A 0R6

REVIEW OF THREE DIMENSIONAL WATER FOG TECHNIQUES FOR FIREFIGHTING

Z. Liu, A. Kashef, G. D. Lougheed and N. Benichou

ABSTRACT

This report provides a review of research into, and application of, a three dimensional (3D) water fog technique for firefighting. The impact of water fog characteristics associated with properties of the nozzle (e.g., droplet size, momentum, flow rate, spray angle and pattern) and discharge techniques (e.g., discharge angle, and discharge duration related to the bursts) on performance of the 3D water fog technique are discussed. Potential use of numerical computer studies to help understand and improve this technique is also reviewed and discussed.

The 3D water fog technique is not designed to replace the direct fire attack but rather to complement existing forms of fire attack in an effort to increase the safety and effectiveness of fire fighting teams. Compared to the traditional straight-stream attack, the 3D water fog technique has advantages in controlling steadily growing fires where the space can still be entered, but where the seat of the fire cannot be attacked directly. It has also been used for offensive attack to control flashover. However, there is not sufficient research to evaluate its capabilities in other fire scenarios, such as reducing the likelihood of backdraft, and in controlling fire threats in low visibility scenarios. Research on the effects of the nozzle type, application techniques and fire conditions on the performance of the 3D water fog tactic is also very limited. Further research efforts, including both experimental and numerical studies, can help firefighters understand how to most effectively use this technique and also help to improve its performance in firefighting.

Key Words: 3D water fog technique, gas cooling, fire suppression, firefighting, computer simulation

INTRODUCTION

Interior structure firefighting is now becoming more challenging and dangerous than in the past. Many new buildings, such as commercial centers, galleries, exhibition halls and warehouses, are becoming bigger and more complex. These buildings have big open indoor areas containing extensive volumes of combustion gas once a fire occurs. Also, the extensive use of new materials for building construction and contents can result in increasing smoke productions and more rapid flame development once they ignite. The use of new construction technologies, such as the widespread installation of double-glazed, energy-efficient windows, that allow hot smoke to be contained in the room for an extended period, also adds to the challenges for firefighting [1, 2].

Conventionally, water has been used as an efficient agent in controlling a fire and minimizing its spread. There are two traditional fire-attack methods, namely, direct and indirect attack. In the direct attack (DIR), water is applied to the base of the fire using either a straight stream or narrow-angle fog as firefighters advance into the immediate fire area. In the indirect attack, water is applied to hot surfaces to produce steam to smother the flames. The direct attack method is preferred for an incipient or growing unobstructed fire, whereas an indirect attack is preferred for a post-flashover/fully developed fire. However, for a wide range of interior fires, these approaches may not be effective. Some of these fire scenarios where these approaches are not effective include:

- Situations where the unburned smoke is very hot and near to its auto-ignition point. Without cooling or diluting the unburned smoke gases first, the smoke may suddenly auto-ignite and develop into a full fire involving all the combustible elements within the compartment (flashover) [3].

- Fire scenarios involving the occurrence of backdraft in which a fireball or a blast wave may be generated in an under-ventilated compartment when fresh air is introduced as firefighters enter the room to initiate fire attack.

- Interior fires in which flames are shielded by obstructions, where there are multi-fire sources scattered within the compartment, or where the visibility of the room is very poor and the fire is obscured [4]. The conditions (heat, smoke and fire gases) in these scenarios may not prevent initial entry into the fire compartment. However, firefighters need extra time to locate and attack the fires during which the fire may continue to grow rapidly, potentially resulting in flashover conditions.

For these fire scenarios, the hot gases must first be cooled down or diluted in an attempt to prevent potential flashover or backdraft, to improve and maintain tenable conditions for firefighters, and to allow the firefighters more time to locate and extinguish the fire.

In structural firefighting, both straight water stream and narrow-angle fog are used to cool hot gases and to maintain control of the fire environment. However, using these tactics may create a massive amount of hot steam, causing burns and discomfort to the firefighters. It may also disrupt the smoke layer, and cause it to descend, further reducing visibility and increasing discomfort and risk to the firefighters [3, 4].

To overcome these disadvantages, a three dimensional (3D) water fog technique has been developed and its use has been investigated in fires since the 1980s [5]. This technique uses a combination fog nozzle to inject fine water droplets into overhead gas layers in a series of short bursts or “pulses”. The objective is to suspend fine water droplets in the smoke layer to cool, inert and dilute unburned hot gases, bringing them outside their flammability range in an attempt to prevent or quench subsequent ignitions.

The 3D water fog technique is not designed to replace the direct fire attack but rather to complement existing forms of fire attack in an effort to increase the safety and effectiveness of firefighting teams. Over the last two decades, several fire authorities in Sweden, UK, Australia, Spain and the US Navy (for firefighting onboard surface ships) have officially adopted the 3D water fog techniques for firefighting. This approach is also under review in the USA, Holland, Germany and France [3].

The use of the 3D water fog technique for gas cooling or fire suppression is a complicated process. A water fog spray is a two-phase absorbing, emitting and scattering medium, including a droplet poly-dispersion (liquid) phase and water vapour (gaseous) phase. The interaction between the water droplets and the hot gas occurs through momentum and energy exchanges. The nature of this interaction is affected by many factors, including the characteristics of the water fog associated with properties of the nozzle, application techniques, fire conditions and compartment geometry. With the current limited understanding of the 3D water fog technique, there are concerns in using it for firefighting [2, 3]. These concerns include:

- its capability of controlling the fire threat, compared to traditional straight stream attack;
- the possible disruption of the thermal balance in the compartment;
- the possible generation of a large amount of hot steam that produces a burn hazard to firefighters, and;
- The performance of the 3D water fog technique is determined by many factors and extensive training is required.

While advocates and users of the 3D water fog technique have attempted to respond to these concerns, only very limited experimental studies have been undertaken due to the complexity of the problems. To date, the 3D water fog technique is not widely used for firefighting, especially in North America.

This report reviews the existing literature dealing with this technique and identifies research needs. The impact of water fog characteristics associated with properties of the nozzle (e.g., droplet size, momentum, flow rate, spray angle and pattern) and discharge techniques (e.g., discharge angle, and discharge duration related to the bursts) on the performance of the 3D fog technique is also discussed. In addition, the potential use of numerical computer studies to help understand, use and improve this technique is also discussed.

DEVELOPMENT OF 3D WATER FOG TECHNIQUE

There is renewed interest in the use of 3D water fog techniques for gaseous-phase suppression of structural fires. It was first investigated in Germany in the 1950s but it failed to progress beyond theory. In the 1980s, firefighters in Sweden began to test this technique in real fires after a major flashover fire killed two firefighters. By 1984, this technique was being used by firefighters in London, England and throughout Sweden. Since then, several firefighting manuals and articles have been published describing this technique [5, 6, 7]. These early investigations focused mainly on the evaluation of the performance of the 3D water fog technique in fires, and no detailed study on its effectiveness in gas cooling, generation of steam or disruption of thermal balance in the fire compartment was reported.

Starting from 1994, the Naval Research Laboratory [4, 8] carried out a series of full-scale fire tests to determine the benefits and drawbacks of using traditional straight-stream attack versus the 3D water fog technique to control a steadily growing fire threat. These tests were conducted within the confines of the Navy ship ex-USS SHADWELL with a fire space of approximately 110 m^3. Cooling effectiveness, generation of steam and disruption of the thermal balance in the fire space were investigated and compared to the straight-stream tactic.

There were three types of fire threats investigated in the test series. The first one was a steadily growing fire that had multiple fire source locations dispersed about the fire compartment. The overhead gases, with visible flame, were very hot and the temperature ranged from 400 to 600^{o}C, creating flashover or near flashover conditions in the space. At the same time, obstructions were placed between the fire sources and the entry point to the fire compartment, forcing the attack team to advance well into the space, under severe conditions, to directly attack and extinguish the fires. The second fire threat was very similar to the first one but no obstructions were placed within the fire space. The third fire threat was a low visibility fire, posing a significant challenge in locating the fire sources. In all cases, the fire was fought using a 38 mm handline equipped with a 360 L/min nozzle. For the 3D water fog attack, the nozzle was set to produce a 60^o fog pattern and the stream was discharged upwards at a 45^o angle into the smoke layer. For traditional fire attack, the nozzle was set either for a straight stream or 30^o spray angle position with a water directing at the fire or the smoke layer.

For the obstructed fire scenarios, the 3D water fog technique had advantages over the traditional stream attack in controlling the fire threat. Several bursts (2-3 s in duration) of water fog were sufficient to attack and control the overhead fires. The smoke temperature was quickly reduced by 200-250^{o}C and then continued to cool. With the straight stream tactic, the overhead temperatures were reduced initially but quickly returned to their original level. The steam generated with the 3D water fog technique was less than that produced using a straight stream or narrow angle fog. It was a moist, “sweaty” type of steam, resulting in no burns to the participants, rather than a hot, penetrating steam. A straight stream or narrow angle fog directed into the smoke layer, however, resulted in excessive amounts of hot, penetrating steam, which caused burns to the hands, wrists, face, neck and backs of the firefighters. Heat flux data indicated that there was no significant disturbance in the thermal balance in the room with the 3D water fog technique, while the thermal balance was disturbed in several tests with the traditional straight stream attack.

For non-obstructed fire scenarios, full-scale tests showed that neither the 3D water fog technique nor the traditional straight stream had a clear advantage. The tenability and extinguishment benefits provided by using the 3D water fog technique were offset by the ability to mitigate the threat quickly by applying water directly to the fire using the straight stream approach.

For the low visibility scenario, only one test was conducted using each approach. As such, the performance of the two approaches could not be compared but it was observed there was no disadvantage in using the 3D water fog technique in this scenario.

In order to compare the potential effect on firefighter heat strain and the risk of heat-related injuries, a further series of fire tests were conducted by the USA Naval Research Laboratory (NRL) [8] using the obstructed fire scenario. The heart rate and temperatures (rectal, chest, arm, hand, finger, thigh, foot and big toe) of the firefighters were measured during firefighting. Test results showed that use of the 3D water fog technique reduced smoke temperatures and controlled thermal balance, allowing firefighters to locate and extinguish the fires more rapidly. Compared to the straight stream attack, the firefighting time using the 3D water fog technique was approximately 2 minutes shorter. This resulted in a lower level of heat strain for the firefighters. The heart rate and body temperatures of the firefighters using the 3D water fog technique were lower than those using the traditional straight spray attack.

Based on the test results, the Naval Research Laboratory (NRL) [4] recommended that the benefits of the 3D water fog technique as an offensive tactic should be addressed in the firefighting manuals, and that it could be used to control steadily growing fires where the space can still be entered, but where the seat of the fire cannot be attacked directly.

Schnell [9] reported flashover training conducted in Sweden. The 3D water fog technique is one of the steps in their offensive attack. After an initial 5 s discharge with two fast showers of water, the 3D water fog technique was used to cool the combustion gases and to extinguish the fires. The nozzle was set to provide a 60^o fog pattern and the fine droplets in short bursts or "pulses" were discharged upwards at a 45^o angle into the smoke layer with a maximum distance before the droplets hit the ceiling. Subsequently, a low water flow rate (100 L/min) was used to cool the walls and ceiling, reducing the production of combustion gases. This also reduces the generation of steam. Finally the fire was put out using direct water application. Firefighters in Sweden have been trained to use this approach since 1986. It was reported that flashover was more efficiently controlled [14].

Some reports and articles have also claimed that the 3D water fog technique can reduce the likelihood of backdraft [3] but there has been no research to validate this claim. However, other studies have shown that the injection of water spray into a fire compartment can suppress backdraft [10, 11]. Gottuk, et al [11] reported that water injection can be used as a mitigating tactic to suppress a diesel fuel backdraft. The ability to mitigate and eliminate backdraft by using water spray increases with the amount of water injected. The severity of the backdraft can be reduced from a blast wave with a fire ball to the complete prevention of ignition. The backdraft suppression by water spray occurs primarily by means of diluting the atmosphere and reducing the fuel mass fraction rather than by a thermal cooling mechanism. However, Gottuk et al [11] did not report how and what kind of water spray was used for mitigating the backdraft, and the effect of droplet size and other spray characteristics (e.g., spray angle, distance, and velocity) on the effectiveness of backdraft suppression was not investigated.

EFFECT OF WATER SPRAY CHARACTERISTICS AND APPLICATION TECHNIQUES ON PERFORMANCE OF 3D WATER FOG TECHNIQUE

The performances of the 3D water fog technique are generally determined by the nozzle characteristics (e.g., droplet size and velocity, spray angle, and flow rate), and application techniques (e.g., discharge angle, and duration of discharge). When using the 3D water fog technique, the nozzle and application technique are different from those used in the direct and indirect attack methods. However, currently there are very limited experimental studies of their effect on the performance of the 3D water fog technique. In the NRL tests [4], for example, only one 60 degree fog-cone was applied at a 45 degree angle to the smoke layer. The performance of the 3D water fog technique with different spray angle, discharge angle, droplet size, discharge pressure, and flow rate was not studied. In this section, the effect of various parameters on gas cooling, generation of steam and disruption of the thermal balance using the 3D water fog technique is analyzed and compared to the traditional straight stream approach.

In theory, small droplets are more efficient in cooling and diluting the gases than large droplets, because of the larger total surface area available for evaporation and heat extraction. As shown in Table 1, when the droplet diameter is reduced from 1000 μm to 100 μm, the total surface area increases 10 times from 6 m^2 to 60 m^2 for 1 litre of water [12].

Table 1 Variation of Surface Area with Droplet Diameter (1 litre of water)

Droplet size (μm)	1000	100	10
Total number of droplet	1.91×10^6	1.91×10^9	1.91×10^{12}
Total surface area (m^2)	6	60	600

Small droplets also evaporate and absorb the heat more quickly than large droplets. The lifetime of the droplet, t_{life} (s), is mainly determined by droplet size and the surrounding temperature and can be approximately given by Andersson et al [13] for droplets in the range 0.1 – 1 mm:

$$t_{life} = \frac{DL_e \rho}{2K_g \Delta T C_2} \qquad (1)$$

where D is the droplet diameter (m), L_e is the latent heat of water (kJ/kg), ΔT is the temperature difference between a water droplet and the surrounding air (^{o}C), ρ is density of water (kg/m^3), K_g is the thermal conductivity of the surrounding air (W/m.^{o}C), and C_2 is a constant (m^{-1}). Based on Equation (1), the lifetime of droplets in different environmental temperatures is listed in Table 2. It indicates that the lifetime of a droplet is quickly reduced with a decrease in droplet size and an increase in air temperature. When the droplet size is reduced from 1000 μm to 100 μm, the lifetime of the droplet in still air is reduced by 10 times.

Table 2 Variation of Lifetime of Droplets With Temperature

ΔT (^{o}C)	100 (μm)	200 (μm)	300 (μm)	500 (μm)	1000 (μm)
200	0.8 s	1.6 s	2.4 s	4.0 s	8.0 s
300	0.533 s	1.06 s	1.6 s	2.66 s	5.33 s
400	0.4 s	0.8 s	1.2 s	2.0 s	4.0 s
600	0.26 s	0.52 s	0.78	1.3 s	2.6
800	0.2 s	0.4 s	0.6 s	1.0 s	2.0 s
1000	0.16 s	0.32 s	0.48 s	0.8 s	1.6 s

The cooling effectiveness of water spray for hot gases is also determined by the residence times of droplets that are available for absorbing heat from the gas. The longer the residence time, the better the cooling effectiveness of the spray. The residence time of a droplet in the hot gases, t (s), is mainly determined by the initial spray velocity or discharge pressure, droplet size and penetration distance. For a downward spray, it can be estimated by [14]:

$$t = \frac{D\rho_l}{0.33\rho_g U} \qquad (2)$$

where D is droplet diameter (m), ρ_g and ρ_L are air and water density (kg/m^3), respectively, and U is droplet velocity (m/s) and can be determined by:

$$U = \frac{U_o}{\exp\left(\frac{0.33\rho_g L}{D\rho_l}\right)} \qquad (3)$$

where U_o is initial droplet velocity and L is the penetration length (m).

Table 3 shows, based on Equations (2) and (3), the time needed for a droplet to reach the floor under different discharge pressures, when the downward penetration distance is assumed to be 3 m. It indicates that for the same discharge pressure, small droplets have much longer residence times than large droplets. With increase in discharge pressure, the residence time of the droplet is reduced. For a spray discharge, the residence time is further reduced compared with an individual droplet, because the leading droplets impart forward momentum to the surrounding gas, reducing the air drag on the following droplets and resulting in a better overall penetration [15].

Table 3 Times that droplets need to reach the floor (3 m downward penetration)

Discharge pressure (MPa)	5	10	30	68
200	3 s	2 s	1.12 s	0.74 s
300	2.7 s	1.75 s	0.96 s	0.64 s
500	1.16 s	0.77 s	0.41 s	0.27 s
1000	0.86 s	0.57 s	0.3 s	0.2 s

Other important factors in determining the cooling effectiveness of the 3D water fog technique include the amount of water applied, the spray angle and the discharge angle. Increasing the amount of water discharged will result in more water being available for cooling. For a fixed water flow rate, the amount of water applied using the 3D water fog technique is mainly controlled by the duration of the discharge. Increasing the spray angle will increase spray volume and contact area with the hot air, while changes in discharge angle will result in changes in spray penetration distance, the residence time and the spray volume available for gas cooling.

The spray volume discharged from a nozzle is mainly determined by the spray angle and penetration distance and can be approximated using:

$$V = \frac{1}{3}\pi\left(L\tan\frac{\theta}{2}\right)^2 \qquad (4)$$

where θ is the spray angle.

Table 4 shows the variation of spray volume with spray angle and penetration distance. It indicates that the spray volume increases significantly with spray angle and penetration distance.

For a 3 m spray penetration length, the spray volume with a spray angle of 90° is approximately 14 times larger than that with a spray angle of 30°.

Table 4 Spray volume with spray angle and penetration distance

Spray angle (°)	30	60	90
Spray volume (m^3) (2 m of penetration)	0.3	1.39	4.18
Spray volume (m^3) (3 m of penetration)	0.676	3.14	9.42

These analytical results are consistent with experimental results that the 3D water fog technique could be more effective in cooling hot gases than traditional straight or narrow angle attack due to the fine droplets and wide spray angle [4].

However, one concern in the use of the 3D water fog technique is that excess steam may be generated as large amounts of fine droplets are discharged. This can reduce visibility and cause comfort problems to the firefighters.

For overhead cooling, the steam is generated from two sources: water droplets evaporating in the hot gases as they pass through the gas; and water droplets evaporating on hot surfaces, such as the ceiling and walls as they reach and deposit on these surfaces. The amount of steam generated is determined not only by the amount of water applied, but also by spray angle, droplet size, discharge pressure and angle (in relation to the horizontal), as well as the fire conditions, such as fire size, room temperature and geometry. Excess steam may be generated if the droplets are too fine or the spray angle is too large, because of the large surface in contact with the hot gases and the resulting high evaporation rate. However, excess steam may also be generated if the droplets are too big. They will pass through gases to reach and evaporate on the hot surfaces that have higher heating capacity than the gas. In addition, more steam will be generated with the increase in penetration distance or discharge angle as the residence time and spray volume increase.

Use of the 3D water fog technique can control the generation of steam from two aspects. The first one is that a limited amount of water is applied into the overhead gases by using short bursts or “pulses” of water injection. For example, if individual pulses last between 0.1 – 0.5 of a second then the total amount of water discharged for a flow of 360 L/min will be only 0.6 – 3 liter of water. The second aspect of controlling the generation of steam is that the discharge of fine droplets reduces contact between the water and the heated walls and ceiling because the droplets completely evaporate before they reach the hot surfaces.

However, the generation of steam is determined by many factors. Any changes in spray angle, droplet size, discharge pressure, flow rate and burst duration, when using the 3D water fog technique, will result in different cooling effectiveness and generation of steam. No research has been conducted on the effect of spray characteristics and application techniques on the cooling of hot gases and the generation of steam during firefighting.

Another concern with the use of the 3D water fog technique is the possibility of a disruption of the thermal balance in the compartment. With the spray discharge, the compartment is cooled quickly. This may result in a negative pressure in the compartment, increasing the flow of fresh air, and changing the thermal balance in the compartment. At the same time, droplets and entrained air caused by spray discharge may block the movement of smoke and hot gases in the upper layer and redirect them to mix with cooler air in the lower layer, resulting in visibility and comfort problems. In addition, the expanding steam may push smoke, heat, and occasionally fire into unaffected parts of the compartment.

The disruption of the thermal balance by the water spray is also affected by the spray characteristics, amount of entrained air, discharge techniques, fire size and compartment geometry. The spray momentum, M_w, can be expressed as follows:

$$M_w = (m_{wl} + m_{wv} + m_{wa}) \times V_w \qquad (5)$$

where m_{wl}, m_{wv}, and m_{wa} are the mass of liquid-phase water, vapour-phase water and air entrained by the spray, respectively, and V_w is the velocity vector of the spray. The equation shows that the amount of water discharged and spray velocity are important factors in determining the spray momentum. Increasing the amount of water discharged will increase the spray momentum and the amount of entrained air, and generate more steam, leading to a greater disruption of the thermal balance.

Air entrainment caused by water discharge may disrupt the smoke flow or even promote a more intense combustion during the initial stages of water discharge. Previous studies have shown that the amount of entrained air increases with water flow rate, spray angle and distance from the orifice [16, 17, 18]. For a given nozzle size, spray angle and distance from the orifice, it has been shown that the entrainment flow is proportional to the water flow rate. Wider spray angles entrain more air than narrower ones. For example, a 90^o angle nozzle could entrain 5 times more air than a 28^o angle nozzle. However, air entrainment is not particularly sensitive to droplet size.

Proper use of the 3D water fog technique can minimize the disturbance to the thermal balance, as it controls the amount of water applied and the entrainment of air by using short duration discharge, although its spray angle is larger than a straight stream. Any improper nozzle or application techniques using the 3D water fog technique, such as long discharges and long penetration distances, could result in serious disruption of the thermal balance in the fire compartment. The effect of spray characteristics and application techniques on the disruption of the thermal balance, when using 3D water fog technique, has not been systematically studied.

NUMERICAL INVESTIGATION OF 3D WATER FOG TECHNIQUE

While full-scale experiments are very useful in the study of the 3D water fog technique, for a number of reasons they alone cannot provide a comprehensive understanding of the technique. These include the complexity of the interaction between the droplets and the hot gases, the high cost of conducting full-scale experiments, the difficulties associated with visualization during the experiment, and finally, the large number of parameters affecting the process.

With the rapid development of computer technology, the use of Computational Fluid Dynamics (CFD) models to simulate fire development, smoke movement and water suppression is increasing. CFD models are two- or three-dimensional models that divide a domain into many cells. A set of mathematical sub-models is applied to compute the thermal and flow conditions in each cell. These models provide detailed information, which can easily be visualized, about the temporal-spatial fire growth, turbulence, thermal radiation, as well as hot smoke spread in the modeled domain. This information can, in turn, offer insights into the interaction between fires and water droplets, the distribution of spray droplets, the mass and heat transfer between the hot gases and droplets. This can help people to understand and use the 3D water fog technique, and help identify potential improvements.

A number of commercial general-purpose CFD computer programs have been developed. Examples of these include PHOENICS [19], FLOW3D [20], and FLUENT [21]. Several CFD models have been developed specifically to model fire related problems. Among these are BF3D [22], UNDSAFE [23, 24], FDS [25] and JASMINE [26]. The SFPE Handbook of Fire Protection Engineering [26] summarizes the capabilities and hardware requirements for CFD models. However, there has been no numerical study of the 3D water fog technique. Most of the numerical research on the interaction between water droplets and hot gases has been for water sprinkler and water mist applications.

To simulate the 3D water fog technique for gas cooling or suppression, the CFD model should contain a number of modules: fire growth and smoke movement models; water spray or fog model; and gas cooling and suppression models.

The fire sub-model is used to describe the fire growth and hot gases or smoke movement in the compartment. It involves the assembly of the governing equations describing the transport of mass, momentum, and energy by the fire induced flows and the implementation of boundary conditions (domain dimensions, configuration, leakage, fire size, fuel type, etc.). The fire can be simulated either as a heat source (based on empirical formula) in which the heat release rate and species production rates are defined as volumetric sources in control volumes at the fire location [28], or as a flamelet model in which the mass production of carbon-related products is determined by combustion chemistry [29, 30]. For example, the combustion sub-model developed by Lai [29] and Hadjisophocleous et al [30] to simulate fire suppression using water mist, was a global local equilibrium model, where chemical species achieved local thermodynamic equilibrium with a rate determined by a combination of local turbulent mixing and global chemical kinetics. The fuel evaporation rate was calculated from the balance of the radiant energy reaching the fuel surface.

The turbulent effect on the mixing and transport of hot gases can be described by either a standard k-ε sub-model or a large eddy simulation. For typical fires, the small-scale vortices associated with viscous forces with energy dissipated into heat are in the order of millimeters. Thus high grid resolution is vital to properly model these small scales. The majority of CFD models avoid this costly constraint by using turbulence models, e.g. k-e models [28], to approximate the turbulent energy and dissipation produced by the fire. This approach results in a steady-state solution to an averaged version of the flow equations. Another approach, the Large Eddy Simulation (LES) [25], is used to solve the large scales of motion and model the small scales that are assumed to be universal. The LES results in a transient solution to the actual Navier-Stockes equations. Because real turbulent flow situations are inherently transient, LES methods can have an advantage in modeling turbulent fire-induced flows. An example of the CFD-LES method is the Fire Dynamic Simulation (FDS) model developed by the National Institute for Standard and Technology [25].

The water spray sub-model is used to simulate the behaviour of water droplets injected from a nozzle, and the distribution and movement of hundreds of thousands of droplets in a compartment. The liquid atomization is a complex process and currently the initial state of the water spray or mist, such as droplet size, and water density, is mainly determined experimentally and is provided to the water spray sub-model as input parameters [31, 32]. For most current water spray models [32-34], interactions between droplets are not considered and water droplets are treated using a Lagrangian tracking model. Individual droplets are tracked from their point of injection until they evaporate. This approach provides detailed information on the distribution of water droplets in the compartment. However, to consider hundreds of thousands of droplets discharged from a number of sprinklers or mist nozzles, significant computer power is required to simulate the distribution of water droplets in the compartment. For the 3D water fog technique, only one nozzle is employed but its water flow rate is much higher than a sprinkler or mist nozzle, and its discharge approach is also different from the sprinkler and mist nozzle. To model suppression using water fog, a new sub-model based on features of the 3D water fog technique would need to be developed.

The fire suppression or gas cooling sub-model simulates the interaction of water droplets with the hot gas layer, radiation field and the flame. The effect of water spray (droplet size, flow density, momentum) on gas cooling and fire suppression can be determined by considering heat and mass transfers, and momentum exchange between droplets and hot gases or fires. In previous studies on fire suppression by sprinkler and water mist [30-34], the actions of the hot gas on the droplets were introduced through the drag function terms in the droplet momentum equations, and the heat transfer terms in the liquid-phase energy and mass conservation equations. In some studies, the effect of heat sources representing absorption from the radiation field on the droplet temperature was also considered. For example, in the water mist studies carried out by Mawhinney et al [32] and Dembele et al [35, 36], the radiation model utilized a six-flux element coupled to the gas and mist phase to simulate the generation of thermal radiation by the fire and its transport throughout the compartment. The radiation equations included an energy term representing the addition to the radiation field of the black body radiation emitted by the hot gas. This energy source was the driving force for the radiation field. The model assumed that the soot temperature was identical to the gas temperature.

For fire suppression by water droplets, extinguishing criteria were developed based on both thermal cooling and oxygen displacement [29-34]. For example, in the computer model developed by Hadjisophocleous et al [29], fire extinction is assumed when the temperature in any given cell decreased below 800 K.

Over the years, CFD computer models have been successfully used to study fire suppression by water sprinkler and mist. For example, a European research project ASTRRE (Atténuation des Sources Thermiques Radiatives par Rideaux d'Eau) [35, 36] developed a predictive computer model for the design of water spray thermal barriers for various large-scale fire hazards. Mawhinney et al [32] developed a CFD model to aid in optimizing the design of water mist suppression systems as a possible replacement for halon-based fire suppression systems. This work was part of the FIRE Detection and Suppression Simulation (FIREDASS) project, a European Union funded BRITE/EuRam research project set up to address the requirement for alternative suppression and improved detection system. The Naval Research Laboratory has developed and applied a numerical model to study the combustion of methane-air diffusion flames and their inhibition by water mist on a laboratory scale [37]. A number of studies using CFD models for full-scale fire suppression research have been carried out by Hadjisophocleous, et al [29, 31, 33, 34, 38]. They studied the effect of water mist on liquid pool fires in open spaces and in a compartment with various obstacles and a number of mist nozzles (e.g., 24 nozzles), and fire suppression by water mist in an aircraft cabin. The predicted results show good agreement with the corresponding experimental values.

The results obtained with CFD modelling demonstrate that it is a promising tool for exploring the complex physical phenomenon of gas cooling and fire suppression by water droplets. It can extend our current understanding of the interaction between water droplets and fires. The current CFD models, however, require significant computer power for the simulation of hundreds of thousands of droplets in gas cooling and fire suppression. In order to improve the accuracy of CFD modelling, a more comprehensive knowledge of liquid atomization and radiation models is required. Finally, a water spray model would need to be developed to simulate 3D water fog technique.

CONCLUSIONS

- The 3D water fog technique is not designed to replace the direct fire attack as it mainly aims to provide a ‘safe’ approach route to the fire, to improve and maintain tenable conditions for firefighters, and to prevent the likelihood of flashover and backdraft.

- Compared to traditional straight stream or narrow fog techniques, both experimental and analytical results show that proper use of a 3D water fog technique can have a better cooling effectiveness, generate less steam and lead to less disruption of the thermal balance in the smoke layer by using short discharges, fine droplets and wide spray angle.

- The 3D water fog technique has demonstrated advantages in controlling steadily growing fires where the space can still be entered, but where the seat of the fire cannot be attacked directly. It has also been found effective for offensive attack to control flashover. However, there is not sufficient research to evaluate its capability for other fire scenarios, such as preventing the likelihood of backdraft, and controlling fire threats in low visibility scenarios.

- The optimum performance of the 3D water fog technique is determined by the characteristics of the water spray (e.g., droplet size and velocity, spray angle, flow rate, etc.), application technique (e.g., discharge angle, and duration of burst discharge), and fire conditions (e.g., fire size and compartment geometry).

- The impact of nozzle parameters, application techniques and fire conditions on the performance of the 3D water fog technique has not been well studied. There are no guidelines for firefighters in the proper use of this technique. This limits the potential use of 3D water fog technique for firefighting and requires extensive training in the use of this technique.

- CFD computer simulation has the potential to provide visualization and insights into detailed aspects of overhead gas cooling and/or fire suppression by fine water droplets. This will help in understanding and improving the 3D water fog technique. Currently there are no CFD studies on the use of the 3D water fog technique. However, the knowledge and sub-models developed from CFD studies for fire suppression by sprinkler and water mist can be used to help develop a suitable CFD model for the 3D water fog technique. To achieve this, there is also a need to establish a water fog sub-model based on the features of the 3D water fog technique.

REFERENCES

1. Cohn, B. M., “Plastics and Rubber,” Fire Protection Handbook, 18th Edition, National Fire Protection Association, 1997

2. Fredericks, A. A., “Little Drops of Water: 50 Years Later, Part 2,” Fire Engineering March 2000

3. Grimwood, P., “New Wave 3-D Water Fog Tactics: A Response to Direct Attack Advocates,” Fire Engineering, October 2000

4. Scheffey, J. P., Siegmann, C. W., Toomey, T. A., “1994 Attack Team Workshop: Phase II – Full-Scale Offensive Fog Attack Tests,” Naval Research Laboratory, NRL/MR/6180-97-7944, 1997

5. Grimwood, P., “Fog Attack,” Hoffman, S., Ed., FMJ International Publications Ltd., Redhill, Surrey, UK, 1992.

6. Clark, W. E., “Firefighting/Principles and Practices,” Dun&Donnelley Publishing Corporation, New York, 1974

7. International Fire Service Training Association, “Essentials of Firefighting,” Third Edition, Fire Protection Publications, Oklahoma State University, OK, 1992

8. Hagan, R. D., Bernhard, R. D., Jacobs, K. A. Farley, J. R., Ramirez, L. R., Feith, S. J. and Hodgdon, J. A., “Offensive Fog Water Attack Reduces Firefighting Time and Heat Strain During Shipboard Firefighting,” Naval Health Research Center, Report No. 96-22, 1996

9. Schnell, L. G., “Flashover Training in Sweden,” Fire Engineers Journal, Nov. 1996, pp. 25-28

10. Gottuk, D. T., Farley, J. P. and Williams, F. W., “The development and mitigation of backdraft explosions,” Fire Safety Science – Proceedings of the Fifth International Symposium, International Association for Fire Safety Science, 1997, pp. 935-946.

11. Gottuk, D. T., Peatross, M. J., Farley, J. P. and Williams, F. W., “The development and mitigation of backdraft: a real-scale shipboard study,” Fire Safety Journal 33 (1999), 261-282.

12. Liu, Z. and Kim, A. K., “A Review of Water Mist Fire Suppression Systems – Fundamental Studies,: J. of Fire Protection Engineering, 10 (3), 2000, pp. 32-50

13. Andersson, P., Arvidson, M. and Holmstedt, G. “Small scale experiments and theoretical aspects of flame extinguishment with water mist,” Lund Institute of Technology, Lund University, Report 3080, May 1996

14. Williams, A., “Combustion of Spray of Liquid Fuels,” Elerk Science, London, 1976.

15. Hayes, W. D., “Literature Survey on Dropsize Data, Measuring Equipment, and a Discussion of Drosize in Fire Extinguishment,” NBSIR 85-3100-1, July 1985

16. Rasbash, D. J. and Stark, G. W. V., “Some Aerodynamic Properties of Sprays,” Chemical Engineer, Dec. 1962.

17. Heskestad, G., Kung, H-C. and Todtenkopf, N. F., “Air Entrainment into Water Sprays and Spray Curtains,” FMRC No. 22533, June 1976

18. McQuaid, J., “Examination of Entrained Air Flow Rate Data for Large Scale Water Spray Installations,” HSE Technical Report, No.1, 1978

19. N.C. Markatos, M.R. Malin, and G. Cox, “Mathematical Modeling of Buoyancy-Induced Smoke Flow in Enclosures,” Internastional Journal of Heat and Mass Transfer, 25, No. 1, pp. 63-75, 1982.

20. A.D. Burns, D. Ingrams, I.P. Jones, J.R. Knightly, S. Lo, and N.S. Wilkes, “FLOW3D: The Development and Applications of Rebase,” Harwell Report AERE/R/12693, 1987.

21. B. Hutchings, “Solution of Natural Convection Problems Using Fluent,” Fluent Users Newsletter, 1, No. 1, pp. 6, 1986.

22. R.G. Rehm and H.R. Baum, “The Equations of Motion for Thermally Driven, Buoyant Flows,” Journal of Research og the NBS, 83, pp. 297-308, 1978.

23. K.T. Yang, J.R. Lloyd, A.M. Kanury, and K. Satoh, “Modeling of Turbulent Buoyant Flows in Aircraft Cabins,” Combustion Science and Technology, 39, P. 107, 1984.

24. Y.T. Yang and L.C, Chang, “UNDSAFE-I: A computer Code for Buoyant Flow in an Enclosure,” NBS-GCR-77-84, 1977.

25. McGrattan, K.B., Baum, H.R. Rehm, R.G., Hamins, A. and Forney, G.P, “Fire Dynamics Simulator – Technical Reference Guide”, NISTIR 6467, January 2000, National Institute of Standards and Technology, U.S.A.

26. G. Cox and S. Kumar, Combustion Science and Technology, 52, p. 7, 1987.

27. SFPE Handbook of Fire Protection Engineering, Second Edition, 1995

28. Lai, M.K., “CFD Analysis of Liquid Spray Combustion in a Gas Turbine Combustor”, the International Gas Turbine & Aeroengine Congress & Exhibition, Florida, June 2-5, 1997.

29. Hadjisophocleous, G. V., Kim, A. K. and Knill, K., “Physical and Numerical Modeling of the Interaction Between Water Sprays and a Fire Plume,” 8th International Symp. On Transport Phenomena in Combustion, San Francisco, CA, 1995

30. Forney, G.P. and McGrattan, K. B., “Computing the Effect of Sprinkler Sprays on Fire Induced Gas Flow”, International Conference on Fire Research and Engineering, Orlando, FL., 10-15, 1995.

31. Hadjisophocleous, G.V., Kim, A.K. and Knill, K., "Modelling of a Fine Water Spray Nozzle and Liquid Pool Fire Suppression," International Conference on Fire Research and Engineering, Orlando, FL, 1995, pp. 1-6.

32. Mawhinney, R.N., Grandison, A.J., Galea, E.R., Patel, M.K., and Ewer, J., “The development of A CFD Based Simulator for Water Mist Fire Suppression Systems: The development of the Fire Submodel”, Journal of Applied Fire Science, Vol. 9, No. 4, pp 377-345, 1999.

33. Hadjisophocleous, G.V. and Knill, K., "CFD Modelling of Liquid Pool Fire Suppression Using Fine Water Sprays," Annual Conference on Fire Research, Gaithersburg, MD, 1994, pp. 71-72.

34. Hadjisophocleous, G.V., Cao, S. and Kim, A.K., "Modelling the Interaction Between Fine Watersprays and a Fire Plume," Fourth International Conference on Advanced Computational Methods in Heat Transfer, Udine, Italy, July 1996.

35. S. Dembele, A. Delmas, and J.F. Sacadura, “Water Sprays Protection from Fire Thermal Radiation Hazards.” Journal of Heat Transfer 119, 1997.

36. S. Dembele, A. Delmas, and J.F. Sacadura, “Water Sprays Protection from Fire Thermal Radiation Hazards.” 9th International Symposium on Loss Prevention and Safety Promotion in the Process Industries, Barcelona, 4-8 May 1998.

37. Prasad, K., Li, C., Kailasanath, K., Ndubizu, C., Ananth, R. and Tatem, P., “Numerical Modelling of Fire Suppression Using Water Mist, 1. Gaseous Methane-Air Diffusion Flames,” Naval Research Laboratory, NRL/MR/6410-98-8102, Jan., 1998.

38. Hadjisophocleous, G.V. and Cao, S., "Numerical Simulations of Aircraft Cabin Fire Suppression," 88th Symp. of the Propulsion and Energetics Panel on Aircraft Fire Safety, Oct. 1996.