Available online at www.sciencedirect.com
ScienceDirect
Proceedings of the Combustion Institute 35 (2015) 2723–2729
Proceedings of the Combustion Institute
www.elsevier.com/locate/proci

Characterization of interactions between hot air plumes and water sprays for sprinkler protection

Xiangyang Zhou *

FM Global, 1151 Boston-Providence Turnpike, Norwood, MA 02062, USA

Available online 22 October 2014

Abstract

One important aspect of the complex sprinkler protection process is the interaction between the water spray and the fire plume. In order to provide suitable data for the development and validation of a LES-based fire protection models, such as FireFOAM, a series of small-scale experiments were conducted to examine the interaction of hot air plumes and water sprays through combined gas–liquid velocity and droplet size measurements. Laser-based particle image velocimetry (PIV) was used to acquire the spatially-resolved velocity data; and a shadow imaging system (SIS) was used to measure the water droplet size and volume flux. Hot air plumes with three convective heat release rates (1.6, 2.1 and 2.6 kW) were selected to interact with a water spray at a discharge rate of 0.084 Lpm. The velocity field of the hot air plume and ceiling flow with/without water spray, the droplet size and volume flux of water spray with/without hot air plume were measured. The interaction between the hot air plume and water spray was characterized by the location of the interaction boundary with the momentum ratio of the hot air to that of the spray. The results showed that that the momentum ratio and the evaporation effect due to hot air on the water droplets played a significant role to change the interaction structure and the ceiling layer pattern.

Keywords: Fire plume; Sprinkler spray; Interaction; Flow velocity; PIV

1. Introduction

Water sprays have been one of the most reliable and effective methods to control or suppress a fire. One important aspect of the complex fire protection process is the interaction which occurs between the droplet spray and the turbulent buoyant gas flow induced by a fire. When a downward water spray is injected above a fire source, the droplet undergoes momentum transfer (due to drag force), mass reduction (due to water evaporation) and heat transfer (due to droplet heat-up and evaporation). The trajectories of falling droplets are modified by the upward fire plume so that only a part of the spray can penetrate through the plume to the top of the burning fuel, and a part of the spray is deflected back toward the ceiling, finally falling to the floor well away from the fire source. In addition to the modification of the upward fire plume, there is also an impact on ceiling-jet velocities and temperatures induced by the droplet spray. The outcome of this interaction determines how the droplet size and momentum of the spray, the momenta of the flames and hot plume, and the evaporation of spray in the flames influence the cooling of the fire

* Fax: +1 (781) 255 4024.
E-mail address: xiangyang.zhou@fmglobal.com

http://dx.doi.org/10.1016/j.proci.2014.05.078
1540-7489/

plume, and the degree of water penetration through the fire plume to the burning fuel surface.

The fire research community has long been interested in the experimental measurement and numerical modeling of the interaction between a fire plume and a water spray from a sprinkler. The actual delivered density (ADD) test apparatus was developed at FM Global to characterize and evaluate the capability of sprinkler sprays to penetrate the fire plume of a rack storage commodity arrangement by collecting the water in pans placed beneath a fire [1]. When the ADD is greater than the RDD (required delivered density), fire suppression is expected to occur. The ADD apparatus has provided much useful data; however, little experimental data are available to detail how the characteristics of a fire plume and a droplet spray change during the interaction. Most of the knowledge of this interaction was gained through computer modeling [2–6]. A new computational code, FireFOAM, based on the OpenFOAM framework has been developed at FM Global to further the numerical modeling of water-based fire protection including turbulent reactive flow [7], solid fuel pyrolysis [8], soot radiation [9], water transport [10] and sprinkler sprays [11].

This work was purposed to extend existing studies of the interaction between the hot air plumes and water sprays through measurements of gas–liquid two-phase velocity fields and droplet size variation in a series of small-scale experiments. A diagnostic technique of laser-based particle image velocimetry (PIV) was used for acquisition of spatially-resolved velocity data in the gas–liquid interaction field. This measurement was motivated by the need to develop and validate numerical simulation tools for such flows. New numerical techniques, such as Large Eddy Simulation (LES), have combined spatial and temporal resolution that cannot be fully validated by traditional point-measurement techniques, in which data are not spatially correlated. The variation of water droplet size and volume flux in the interaction field was measured by a shadow imaging system (SIS). The goals of this work were to examine the characteristics of turbulent hot air plume interacting with the water sprays through combined gas–liquid velocity and droplet size measurements, and to use the data for the development and validation of LES-based fire protection model. The hot air plume experiments were selected because the generation of turbulence due to buoyancy and water spray can be studied but the complexities of combustion chemistry are removed. The region of interest in this study was the velocity field of hot air plume and ceiling flow with/without spray, and droplet size and volume flux of water spray with/without hot air plume.

2. Experimental setup

A small-scale plume-spray test facility was set in a laboratory with dimensions of 8 × 9 m and 4 m in height. Figure 1 shows the schematic of the experimental setup, which mainly comprises an upward air flow, a downward water spray and a horizontal ceiling. The four measurement windows (or field of view) illustrated in the figure were used for the PIV measurements. The ADD test apparatus [1,4] was referenced to design the current small-scale test facility.

The hot air source was designed to be a circular nozzle with 72 mm in diameter. The origin of a two-dimensional (2D) cylindrical coordinate system (r–z) was located at the center of the top of the nozzle, where r is the radius from the centerline and z is the height above the nozzle. Hot air was forced into the nozzle through a Leister electric air heater and a tube. The tube was filled with metal screens to make the exit velocity profile uniform. Several fire sizes were simulated by changing the convective heat release rates (HRRs) of the hot air. The convective HRR was calculated from the air flow rate and the temperature rise above ambient. The vertical exit velocity was up to 5.0 m/s, and the exit air temperature was up to 205 °C. The hot air temperatures were measured using seven type K thermocouples (28 Gage).

A Delavan CT-1.5–30 B full cone nozzle was selected to discharge the water spray. This spray nozzle is rated to flow 0.08 Lpm of water at 690 kPa with a 30° initial spray angle. Figure 1 shows that the spray nozzle was installed at z = 560 mm above the hot air nozzle and 30 mm below the center of an aluminum ceiling plate. The ceiling plate size was 1.22 × 1.22 m with 3 mm thickness. Water flow to the nozzle was accomplished by pressurizing a water storage tank with dry grade nitrogen. The nitrogen supply pressure was regulated with a two-stage regulator and the tank pressure was monitored with a pressure

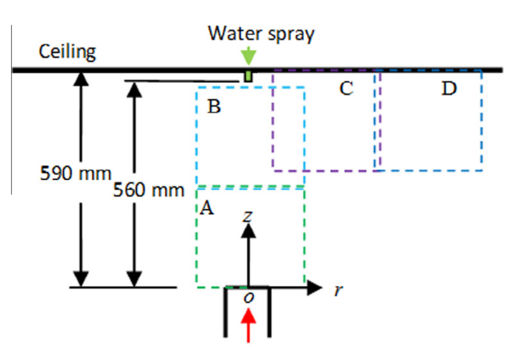

Fig. 1. Schematics of experimental setup: upward air flow, downward water spray, horizontal ceiling and four measurement windows (A, B, C and D) used for PIV measurements.