Fire Safety Journal 96 (2018) 143–152

Contents lists available at ScienceDirect

Fire Safety Journal

journal homepage: www.elsevier.com/locate/firesaf

Numerical modelling of the interaction between water sprays and hot air jets - Part II: Two-phase flow simulations

Tarek Beji *, Setareh Ebrahimzadeh, Georgios Maragkos, Bart Merci

Department of Flow, Heat and Combustion Mechanics, Ghent University-UGent, B-9000, Ghent, Belgium

ARTICLE INFO

Keywords:
CFD
Spray dynamics
Spray-jet interaction

ABSTRACT

The paper presents a comprehensive set of numerical simulations performed to examine the current Computational Fluid Dynamics (CFD) capabilities in the prediction of the interaction of a water mist spray with a vertical upward jet of hot air within an Eulerian-Lagrangian framework. The experimental tests considered herein are described by Zhou [Proceedings of the Combustion Institute, 2015]. The spray is a 30° full cone water mist spray emerging from a nozzle that delivers a water flow rate of 0.084 lpm at a pressure of 750 kPa. The vertical jet of hot air at 205°C is issued from a 72 mm-diameter nozzle placed at 560 mm below the water spray nozzle. Three exit velocities of 3.3, 4.2 and 5.3 m/s were examined. Gas phase simulations (described in the companion paper, Part I) have allowed to determine a set of parameters (e.g., cell size of 4 mm and modified Deardorff model for the turbulent viscosity) that are suitable for the water mist spray simulations. Moreover, it is shown here that a prescribed complex spray pattern with a full discharge angle of 60° is required in order to match water spray profiles in the nozzle near-field. The three regimes of spray-jet interaction (i.e., water spray dominated, vertical jet dominated or equal influence of the spray and the vertical jet) are qualitatively well captured by the numerical simulations. However, the location of the interaction boundary is underestimated by up to 26%. This could be partially attributed to modelling aspects related to, for example, turbulent dispersion or turbulence inflow conditions of the droplets. Uncertainties in the experimental measurements must also be considered.

1. Introduction

Water sprays are known to be an efficient means for fire control and suppression. The interaction of a water spray system with a fire occurs at many levels. For instance, flames in direct contact with water (in sufficient amounts) are extinguished. Another important aspect is spray surface cooling: when liquid droplets reach the surface of solid materials exposed to thermal radiation from the fire and to convective heat transfer from the hot gases, the surface temperature is kept low thanks to evaporative cooling; potential ignition is therefore inhibited [1]. A third aspect is related to the interaction of a water spray with smoke. Typically, the induced smoke cooling and the entrainment of cool air into the water spray envelope cause a downward smoke displacement [2]. The hot gases can be confined in the immediate vicinity of the fire. The downward motion of the spray may also act as a water curtain [3] [4]. If the water spray is applied directly above a smoke plume issued from a fire, the extent of the penetration of the former through the latter is an important parameter in assessing the level of fire control and suppression [5]. The latter configuration, referred to in the literature as the spray-plume interaction, is the configuration of interest in this work.

The spray-plume interaction has been investigated experimentally in Ref. [6] for fires generated via heptane spray nozzles and with convective heat release rates of 0.5 MW, 1.0 MW and 1.5 MW. The sprinklers used were early suppression fast response (ESFR) sprinklers positioned at a ceiling clearance of 3 m from the fire source and delivering flow rates of 1.88 l/s up to 6.23 l/s. The experimental data remained though limited to water accumulation measurements using buckets positioned at the level of the water source. These measurements are referred to as Actual Delivered Density (ADD) measurements and provide an estimate of the water flux that is actually penetrating the fire plume. In order to provide a more detailed characterization of the spray-plume interaction, Schwille et al. [7] carried out experiments in which 5, 15 and 50 kW methane fires were exposed to a spray positioned at 1.5 m above and delivering flow rates that ranged from 6 to 106 L/min. More specifically, the extent of the interaction region has been associated with significant temperature fluctuations which are correlated with high levels of fluctuations in the infrared (IR) intensity.

* Corresponding author.
E-mail address: tarek.beji@ugent.be (T. Beji).

https://doi.org/10.1016/j.firesaf.2018.01.006
Received 27 November 2016; Received in revised form 12 January 2018; Accepted 15 January 2018
Available online 4 February 2018
0379-7112/

The position of maximum fluctuations was used as a measure of the location of the interaction between the fire and the spray. The quality of the experimental data in the spray-plume interaction scenario has been further improved by Zhou [8] who considered the case of an upward hot air jet (with vertical velocities of 3.3–5.4 m/s and an initial temperature of 205 °C) and a water mist nozzle positioned at 0.560 m above and delivering a flow rate of 0.084 lpm. Detailed measurements were performed using laser-based particle image velocimetry (PIV) to acquire spatially-resolved velocity data and a shadow imaging system (SIS) to measure water droplet size and volume flux. Lately, Link et al. [9] made a remarkable effort in characterizing with more detail the initial spray structure in a spray-plume configuration using a spatially-resolved spray scanning system (4 S). The updraft from a real fire plume has been simulated by a forced air jet with a velocity of around 4 m/s and at ambient temperature. Spray nozzles with k-factor 33.1 lpm/bar$^{1/2}$ operating at 1.38 bar were mounted on a ceiling at 1.5 m above the air-jet outlet. One can clearly see from the four experimental programs described above and listed in a chronological order that experimental research on spray-plume interaction is more and more focused on more controllable conditions (e.g., from liquid and gas fires to hot and then cold air) at laboratory-scale and with more and more advanced diagnostics. Such controllable conditions might not reflect practical fire scenarios but the intent, as explicitly mentioned in Refs. [8] and [9], is to provide data for CFD validation. To the best of our knowledge, there is no well-established theory or semi-empirical correlations on the interaction between sprays and fire plumes given the very wide range of possible fire scenarios (e.g., in terms of power of the fire source or the characteristics of the sprinkler or the water mist nozzle). Thus, the computational fluid dynamics (CFD) tools are believed to be a very good way to deal with such a problem because they allow the study of a large number of cases and scenarios at reduced cost and may result in the development of generalized engineering correlations [5] and/or general design and installation rules [10]. The CFD studies undertaken in Refs. [11] and [12] remained though rather qualitative. For example, in Ref. [11], it is stated that the numerical results should be taken with great caution and that only after the models are validated can the (numerical) study be used as a design tool. It is in this context that we defined the aim of our work as a detailed assessment of a CFD tool, namely the Fire Dynamics Simulator (FDS), based on the experimental data displayed in Ref. [8]. We believe that we are only at the very beginning of the process of understanding the interaction of sprays and plumes in the context of fire suppression, and validated CFD packages will play a very important role in the years to come to build up this knowledge and understanding, because with multiple CFD packages validated, the use of CFD as numerical experiments may become possible and reliable.

The experiments carried out in Ref. [8] have been indeed designed in order to provide detailed and high quality experimental data for the purpose of assessing, improving and, eventually, validating the current CFD capabilities in the prediction of two-phase flows for water/smoke interaction. More specifically, the configuration (addressed in this paper) consists of a ceiling-mounted water spray placed directly above the centre of a hot air jet issuing from a steel tube. Prior to the water spray experiments, three experiments were performed for three hot air jets without a spray [8]. The simulation of these tests has been carried out and described in Ref. [13]. It has been concluded that provided that a good set of modelling options (for example for the turbulent SGS viscosity model or turbulence inflow boundary conditions) is selected, the gas phase flow can be predicted with a relatively high level of accuracy. After characterizing the gas phase flow, the water spray was characterized in Ref. [8] in terms of droplet size and velocity and water volume flow rate at two different elevations from the nozzle (in the near-field and far-field of the spray) without hot air. Finally, the interaction of the three hot air plumes with the water spray was investigated through combined gas-liquid velocity and droplet size measurements.

In Ref. [14], numerical simulations of the experiments described above have been performed with the CFD code FireFOAM, which is based on the open source framework OpenFOAM. Contrarily to the so called isolated thermal plumes that were relatively well predicted (as confirmed later in Ref. [15]), the water spray tests were more difficult to predict in Ref. [14]. The simulations performed in Ref. [14] showed indeed that, in the near-field, the simulated spray is wider than the profile measured experimentally. Furthermore, there were substantial discrepancies in the predictions of the liquid volumetric flow rate and droplet velocity in the core of the spray envelope. Results for the far-field were more encouraging. The simulations of the spray-jet interaction cases revealed that the predicted penetration depths of the spray through the plumes are substantially overpredicted. These discrepancies can nevertheless be partially attributed to experimental difficulties in separating the gas phase and spray velocities [14]. Additional FireFOAM simulations carried out in Ref. [16] have shown that increasing the discharge half-angle form from 15° to 30° (to better match the near-field spray pattern) does improve the results. However, the location of the interaction boundary remains underestimated.

The goal of this work is to improve the prediction of the penetration depth. The predictive capabilities of another CFD code are assessed based on the experiments described in Ref. [8]. This code is the Fire Dynamics Simulator (FDS 6) [17,18].

2. Experimental set-up

2.1. Nozzle parameters

The nozzle used in Ref. [8] is a Delevan CT-1.5-30°B full cone nozzle, which delivers a water flow rate of 0.08 lpm at 690 kPa with a 30° initial spray angle. The water spray nozzle was actually operated in Ref. [8] at $\Delta P_w = 750$ kPa, delivering a water flow rate of 0.084 lpm. The nozzle diameter provided by the manufacturer is $D_{nozzle} = 0.33$ mm.

2.2. Experimental configuration

Fig. 1 shows a schematic of the test configuration examined in this paper. The nozzle is placed at 30 mm below a 1.220 m × 1.220 m aluminium ceiling plate in a centred position. A vertical upward jet of hot air is issued from a 72 mm diameter steel tube at a fixed temperature of 205 °C. Three air velocities were tested: 3.3, 4.2 and 5.3 m/s. Prior to spray/jet interaction experiments, the water spray has been characterized in the absence of the vertical jet of hot air. More details are provided

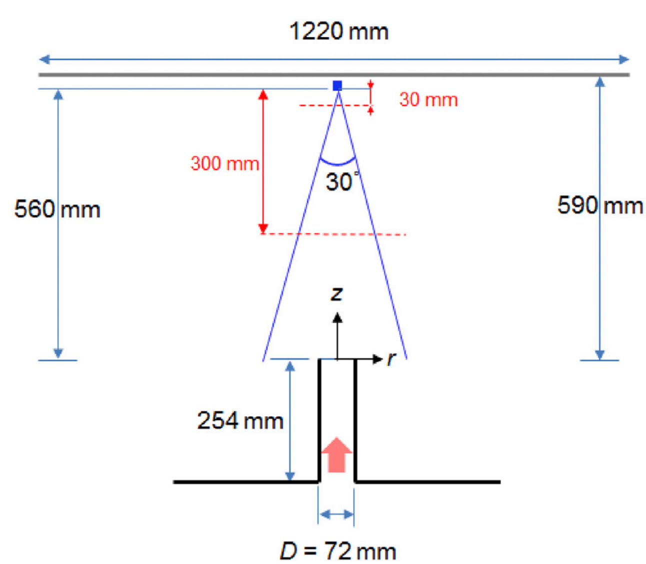

Fig. 1. Schematic image of test configuration (not exactly up to scale).