Fire Safety Journal 95 (2018) 77–86

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 I: Gas phase Large Eddy Simulations

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

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

ARTICLE INFO

*Keywords:*
Computational Fluid Dynamics (CFD)
Turbulence modelling
Vertical jet
Ceiling jet

ABSTRACT

The paper reports a comprehensive set of large-eddy simulations (LES) of a turbulent hot air jet impinging onto a ceiling. The hot air source is a 72-mm diameter circular nozzle with an exit temperature maintained at 205 °C. Three exit velocities have been tested: 3.3, 4.2 and 5.3 m/s, corresponding to Reynolds numbers of respectively 6800, 8600 and 10900 and Froude numbers of respectively 3.9, 5.0 and 6.3. The horizontal aluminium ceiling plate of 1.22 m × 1.22 m has been placed at a distance of 590 mm above the hot air nozzle. This configuration has been examined experimentally by Zhou [Proceedings of the Combustion Institute, 2015] to characterize gas phase conditions prior to experiments which aim at studying the interaction between hot air jets and water sprays. This paper constitutes the first part of a numerical study that aims at assessing the current modelling capabilities of the two-phase flow configuration examined by Zhou [Proceedings of the Combustion Institute, 2015]. The results show that the centerline mean vertical velocity profiles of the vertical jet are predicted with maximum deviations of less than 6% from the experimental data at the condition of an appropriate set-up of the inflow conditions (i.e., geometry of the inlet and turbulence inflow boundary conditions). Furthermore, the best results were obtained with the dynamic Smagorinsky model for the turbulent viscosity. The modified Deardorff results are nevertheless very good given the substantial decrease in computational time (in comparison to the dynamic Smagorinsky model). A good prediction of the vertical jet allowed relatively good predictions of the ceiling jet maximum velocity, boundary layer thickness and Gaussian momentum width with maximum deviations of respectively 20%, 1 mm and 18%. The numerical modelling of the gas phase described in this paper can thus be relied upon in the two-phase simulations described in the companion paper [Part II: Two-phase flow simulations].

## 1. Introduction

Water sprays are known to be an efficient means for fire control and suppression. In conjunction with experimental testing, modelling techniques are continuously being improved in order to be able to evaluate the performance of water spray systems and their ability to create tenable conditions in the fire surroundings. Over the last decades, Computational Fluid Dynamics (CFD) has become a powerful technique that is not only used for academic and research purposes but also as a design tool in many areas of industry, including fire safety engineering. A continuous validation process is nevertheless required to ensure reliable CFD results. This is even more the case for complex two-phase flows characterized by a strong coupling between the gas phase (i.e., hot combustion products) and the liquid phase (i.e., water drops).

The configuration addressed in companion paper (i.e., Part II) consists of a ceiling-mounted water spray nozzle placed directly above the centre of a hot air jet issuing from a steel tube. The experimental campaign described in Ref. [1] aims primarily at providing a detailed and high quality experimental data for the purpose of assessing, improving and, eventually, validating the current CFD capabilities in the prediction of such two-phase flows. Experiments were first performed for a series of three hot air jets (corresponding to three injection velocities) without a spray. Next, a water spray was characterized in terms of droplet size, velocity and water volume flow rate at two different distances 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 is examined through combined gas-liquid velocity and droplet size measurements. Such a stepwise approach is suitable for CFD validation purposes in that it allows assessing first the gas phase and water spray modelling separately. If the level of agreement reached at the end of this stage is deemed high enough then, a potential disagreement between experimental data and numerical results for the spray-jet interaction

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

https://doi.org/10.1016/j.firesaf.2017.11.005
Received 27 November 2016; Received in revised form 21 September 2017; Accepted 13 November 2017
Available online 22 November 2017

could be explained by the need for improvement in sub-models that directly act on the interaction between the two phases, such as the evaporation model.

In this paper (i.e., Part I), the focus is put on the gas phase simulations. In other words, we would like to make sure that the flow field from the hot air is well predicted because any deviation in the hot air momentum at any height will directly impact the spray-plume interaction since the latter is mainly governed by the competition between the momentum of the plume and the momentum of the spray. In Ref. [2], numerical simulations of the experiments described above have been performed with the CFD code FireFOAM, using the Large Eddy Simulation (LES) approach. Results of the so called isolated thermal plumes were however limited to near-inlet velocity and temperature profiles. A more thorough numerical study, also performed with FireFOAM (with the LES approach), has been conducted in Ref. [3] where the focus was put on the influence of the modelling of the turbulent viscosity as well as the turbulence inflow boundary conditions. The obtained results were generally satisfactory. Nevertheless, we observed that the best results were obtained without any subgrid scale (SGS) modelling for a cell size of 4 mm that is not fine enough to have a fully resolved flow. This relatively surprising finding encouraged us to use the Fire Dynamics Simulator (another CFD package that is widely used in the fire safety community [4] [5]) in order to uncover potential differences in numerical dissipation between the two codes. Another point of interest in redoing the exercise with FDS is the treatment of turbulence inflow boundary conditions using the Synthetic Eddy Method (SEM), as opposed to the method of random spots relied upon in FireFOAM.

The general objective of this paper and the companion paper remains though to deliver a complete, comprehensive and careful CFD analysis of the spray-plume interaction with FDS for validation purposes. The comparison with FireFOAM remains for now only at the level of observations of the differences between the two codes. A detailed comparative study (which is out of the scope of the current paper) requires more work and is certainly worth undertaking in the future.

## 2. Experimental set-up

In Ref. [1] a vertical jet of hot air in a quiescent environment is examined. The hot air source is a 72 mm-diameter ($D$) circular nozzle issuing from a 254 mm long steel tube. The hot air exit temperature has been maintained at $T_0 = 205\ ^\circ\mathrm{C}$ at 30 mm above the nozzle exit. Three exit velocities, $w_0$, have been tested: 3.3, 4.2 and 5.3 m/s. A 3 mm-thick horizontal aluminium ceiling plate of $1.22 \times 1.22$ m has been placed at a distance of $H = 590$ mm above the hot air nozzle (see Fig. 1).

Mean velocities (radial and vertical directions) and velocity fluctuations (vertical and horizontal fluctuations, as well as turbulent shear stresses) of the vertical jet and the ceiling jet have been measured using the laser-based Particle Image Velocimetry (PIV) technique. Temperature measurements have not been performed, except at 30 mm above the nozzle exit. More details on the experimental set-up can be found in Ref. [1].

The hot air source can be characterized in terms of Reynolds and Froude numbers calculated as:

$$Re = \frac{w_0 D}{\nu} \tag{1}$$

$$Fr = \frac{w_0}{\sqrt{gD}} \tag{2}$$

where $g$ is the gravitational acceleration and $\nu$ is the kinematic viscosity of hot air (taken as $\nu = 3.5 \times 10^{-5}\ \mathrm{m^2/s}$). The source is further characterized in Ref. [1] in terms of a densimetric Froude number calculated as:

$$Fr_\rho = \frac{w_0^2 T_{amb}}{gD(T_0 - T_{amb})} \tag{3}$$

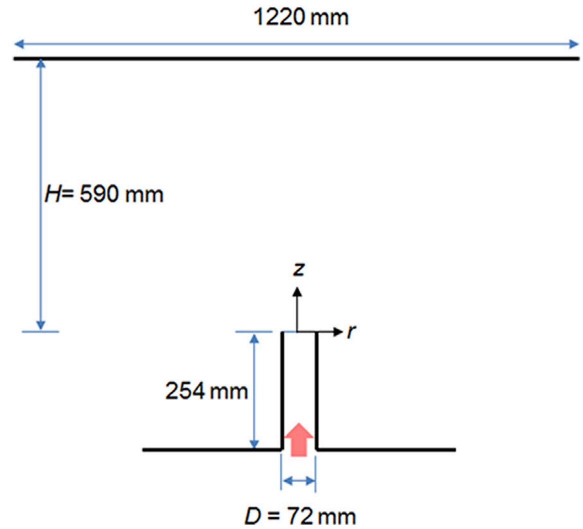


**Fig. 1.** Schematic of the experimental set-up.

where $T_{amb}$ is the ambient temperature taken as $T_{amb} = 20\ ^\circ\mathrm{C}$.

The values of $Re$, $Fr$ and $Fr_\rho$ are displayed in Table 1 for the three cases. It is noteworthy to mention that the obtained values of the Froude number are significantly higher than the ones typically encountered in fires. The obtained vertical jets must then be considered as momentum jets rather than buoyancy-driven plumes. In the general context of fire suppression, this can be seen as a limitation of the current study because the Froude numbers are not representative of fire plumes. Nevertheless, the main purpose in Ref. [1] is to generate data for CFD validation rather than scaling up the results and deriving correlations for fire sources.

## 3. Numerical modelling

The Fire Dynamics Simulator is a CFD code, initially developed for low-Mach number buoyancy-driven flows. However, the latest version has also been successfully applied to high-momentum flows (e.g., $u_0 = 7.2$ m/s and $Re = 5100$ in Ref. [6]). A detailed description of the mathematical modelling in FDS is provided in Refs. [4,5]. The most relevant aspects for the case at hand are recalled here.

### 3.1. Turbulent viscosity models

Turbulence is modeled using the Large Eddy Simulation (LES) technique. Four models are available in Refs. [4,5] for the modelling of the turbulent viscosity: the modified Deardorff (default model), the Smagorinsky (constant and dynamic) and the Vreman model. Two options have been tested in this work, namely the modified Deardorff and the dynamic Smagorinsky model. A third option consists of considering no sub-grid scale (SGS) modelling.

In the modified Deardorff model, the turbulent dynamic viscosity is expressed as:

$$\mu_t = \overline{\rho}\, C_v \Delta \sqrt{k_{SGS}} \tag{4}$$

where $C_v$ is a constant taken as 0.1, $\Delta$ the filter width (taken as the cubic root of the cell volume) and $k_{SGS}$ is the subgrid scale kinetic energy taken

**Table 1**
Reynolds, Froude and densimetric Froude numbers for the three cases.

| | $w_0$ (m/s) | $Re$ | $Fr$ | $Fr_\rho$ |
|---|---|---|---|---|
| Case 1 | 3.3 | 6789 | 3.9 | 24 |
| Case 2 | 4.2 | 8640 | 5.0 | 40 |
| Case 3 | 5.3 | 10903 | 6.3 | 63 |