Fire Safety Journal 86 (2016) 16–31

Contents lists available at ScienceDirect

Fire Safety Journal

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

Large eddy simulation of flame extinction in a turbulent line fire exposed to air-nitrogen co-flow

S. Vilfayeau, J.P. White, P.B. Sunderland, A.W. Marshall, A. Trouvé*

Department of Fire Protection Engineering, University of Maryland, College Park, MD 20742, USA

ARTICLE INFO

Keywords:
Buoyant turbulent diffusion flame
Flame extinction
Fire suppression
Large eddy simulation
FireFOAM

ABSTRACT

The general objective of this project is to support the development and validation of large eddy simulation (LES) models used to simulate the response of fires to the activation of suppression systems. The focus here is on suppression by gaseous agents. The present experimental configuration is a two-dimensional, plane, buoyancy-driven, methane-fueled, turbulent diffusion flame with a controlled co-flow. The co-flow is an air-nitrogen mixture with variable oxygen dilution conditions, including conditions that lead to full flame extinction. Experimental measurements include the global combustion efficiency and global radiative loss fraction. The numerical simulations are performed with a LES-based fire model developed by FM Global and called FireFOAM. In this study, FireFOAM is modified to include a flame extinction model based on the concept of a critical flame Damköhler number and a flame reignition model based on the concept of a critical gas temperature. The numerical simulations are found to successfully reproduce the rapid change that is observed experimentally when exposing the flame to a co-flow with decreasing oxygen strength: the change corresponds to an abrupt transition from a strong flame with a global combustion efficiency close to one to a residual flame with a global combustion efficiency close to zero.

1. Introduction

Computational fluid dynamics (CFD) modeling has emerged over the past two decades as a powerful tool for both research-level and engineering-level projects in the area of fire safety. One of the main technical challenges found in a CFD treatment of compartment fires is the description of partial or total flame extinction, as may occur in under-ventilated fire configurations or in configurations in which a fire suppression system is activated (e.g., systems based on the injection of inert gas or injection of a water spray). This challenge is particularly difficult because flame extinction corresponds to phenomena in which the effects of finite-rate combustion chemistry become a dominant factor: turbulent combustion models used in fire applications typically ignore combustion chemistry (they assume that the rate of combustion is determined by the turbulent rate of fuel-air mixing) and are therefore ill-prepared to simulate flame extinction phenomena.

Available models used to describe flame extinction in fire problems are based on the concepts of a critical flame temperature [1,2] or a critical flame Damköhler number [3–7]. Models based on the concept of a critical flame temperature choose to ignore the importance of chemical time scales and are not consistent with known laminar flame phenomenology [8]. Models based on the concept of a critical flame Damköhler number explicitly or implicitly account for at least one chemical time scale, are consistent with known laminar flame phenomenology, and therefore may be expected to be more accurate [8]. The occurrence of flame extinction is also followed by that of reignition and the modeling of under-ventilated fires or fire suppression requires both an extinction model and a reignition model [4], a difficulty that is generally overlooked in the fire modeling literature.

The objective of the present study is to evaluate the performance of current CFD-based fire models in their treatment of flame extinction and reignition. The study considers an experimental configuration developed at the University of Maryland (UMD) [9,10] and corresponding to a buoyant, turbulent, methane-fueled line fire exposed to air-nitrogen mixtures of variable oxygen strength. The numerical solver is a large eddy simulation (LES) solver developed by FM Global and called FireFOAM [11]. The present study is a continuation of previous work [4]: the extinction/reignition models in Ref. [4] were first formulated using the concept of mixture fraction; these models were then applied to the case of reduced-scale compartment fires with extinction resulting from under-ventilation. The extinction/reignition models in the present study have been re-formulated to a more general framework that does not use mixture fraction; these models are now applied to the UMD line fire experiment characterized by well-

* Corresponding author.
E-mail address: atrouve@umd.edu (A. Trouvé).

http://dx.doi.org/10.1016/j.firesaf.2016.09.003
Received 20 April 2016; Received in revised form 29 September 2016; Accepted 30 September 2016
Available online 13 October 2016
0379-7112/

controlled experimental conditions and a more elaborate instrumentation.

In the following, the experimental configuration is described in Section 2.1; the numerical solver, including a description of the new flame extinction and reignition models, in Section 2.2; and the numerical configuration in Section 2.4. Results on the simulated flame structure are presented in Section 3.1; on grid convergence in Section 3.2; on angular space discretization in Section 3.3; and on flame radiative emissions in Section 3.4. Numerical results are compared to experimental data in Sections 3.5 and 3.6. Section 3.5 uses preliminary experimental data on local variations of temperature and oxygen mole-fraction. Section 3.6 uses recently obtained experimental data on global combustion efficiency.

2. Experimental configuration, numerical solver and numerical configuration

2.1. Experimental configuration

The configuration adopted in the present study corresponds to a buoyant, turbulent, methane-fueled diffusion flame with a controlled co-flowing oxidizer [9,10]. Flames are stabilized above a 5-cm-wide by 50-cm-long slot burner surrounded by a 15-cm-wide by 60-cm-long plate of ceramic fiberboard and further surrounded by a 50-cm-wide by 75-cm-long oxidizer co-flow (see Fig. 1). The plate adds a horizontal component to the flow motion near the flame base and was introduced in previous work with the intent to promote faster transition to fully-turbulent flow conditions [9]. The methane velocity is 6.0 cm/s, corresponding to a mass flow rate of 1.0 g/s, and a nominal total heat release rate of 50 kW for the unsuppressed flame.

In the baseline unsuppressed flame configuration, the oxidizer co-flow is made of pure air; the air velocity is 20.0 cm/s, corresponding to a mass flow rate of 68.5 g/s or approximately four times the stoichiometric requirement of the methane flow. Note that this flow rate is too low to supply all of the gas entrained by the flame and that while the base of the flame is exclusively exposed to the co-flow, the tip region of the flame is exposed to entrained air from the open ambient. While an undesirable feature, numerical tests suggest that this effect remains limited and that the heat release rate associated with the uncontrolled entrained ambient air corresponds to less than 5% of the total heat release rate of the flame [12].

In suppressed flame configurations, the air flow rate remains fixed and nitrogen is added to the co-flow with a mass flow rate between 0–60 g/s; the oxygen mole-fraction in the co-flow, noted X_{O_2}, varies between 21% and 11%. Note that a discrepancy between the numerical and experimental configurations was recently discovered: the numerical configuration follows the protocol described above in which the air mass flow rate is fixed while the nitrogen and total (air plus nitrogen) mass flow rates are variable; in contrast, the experimental configuration follows a slightly different protocol in which both the air and nitrogen mass flow rates are variable while the total mass flow rate is fixed (and equal to 85 g/s). Recent numerical tests suggest that this discrepancy negligibly affects the presently reported results.

An additional small co-flow stream of pure oxygen, called the oxygen anchor, is introduced along the length of the burner to strengthen the base of the flame and oppose liftoff extinction [9]. The velocity of the oxygen anchor is 1.2 cm/s, corresponding to a mass flow rate of 0.08 g/s, and a nominal anchor-limited total heat release rate of 1 kW or 2% of the size of the unsuppressed flame.

Flame suppression is characterized through a variety of diagnostics including measurements of the global combustion efficiency using CO_2 generation and O_2 consumption calorimetry [10], measurements of the global radiative loss fraction using a heat flux transducer combined with time-resolved infrared camera imaging and a multipoint radiation source model [9], measurements of local gas temperature using exposed-junction, 1.0 mm bead-diameter K-type thermocouple probes (uncertainty ± 2 K; response time ~3 s) and measurements of local oxygen mole-fraction using a sampling probe connected to a Servomex 540E paramagnetic oxygen-analyzer (uncertainty ± 1250 ppm; response time ~5 s) [9]. For calorimetry measurements, combustion products are collected in an exhaust duct, where a gas sampling system provides measurement of the mole-fractions of O_2, CO_2, CO and H_2O. From these measurements, the global heat release rate (± 1.5 kW) is derived via mass conservation analysis (for a detailed description of this measurement, the reader is referred to Ref. [10]). Combustion efficiency (± 3%) is determined by dividing the calorimetry-derived heat release rate by the nominal total heat release rate of the flame (50 kW), defined as the product of the measured fuel mass flow rate times the theoretical heat of combustion. The combustion efficiency is a function of the oxygen mole-fraction in the co-flow. Global flame extinction is observed at $X_{O_2,ext} = 12.2\% \pm 0.2\%$.

2.2. Numerical solver

FireFOAM [11,13,14] is based on OpenFOAM [15], an open-source general-purpose CFD software package. FireFOAM is an object-oriented, C++ based, second-order accurate, finite volume solver with implicit time integration; PISO and SIMPLE types of solution procedures are used to couple sequentially-solved equations; the solver features advanced meshing capabilities (the mesh geometry is a structured or unstructured polyhedral mesh); it also features a massively parallel computing capability using Message Passing Interface (MPI) protocols.

FireFOAM uses a Favre-filtered compressible-flow LES formulation and provides a choice between several modeling options for the treatment of turbulence, combustion, and thermal radiation. In the present study, subgrid-scale (SGS) turbulence is described using the one-equation eddy viscosity model (a model based on solving a transport equation for SGS turbulent kinetic energy or TKE) [16]. Combustion is described using the classical concept of a global

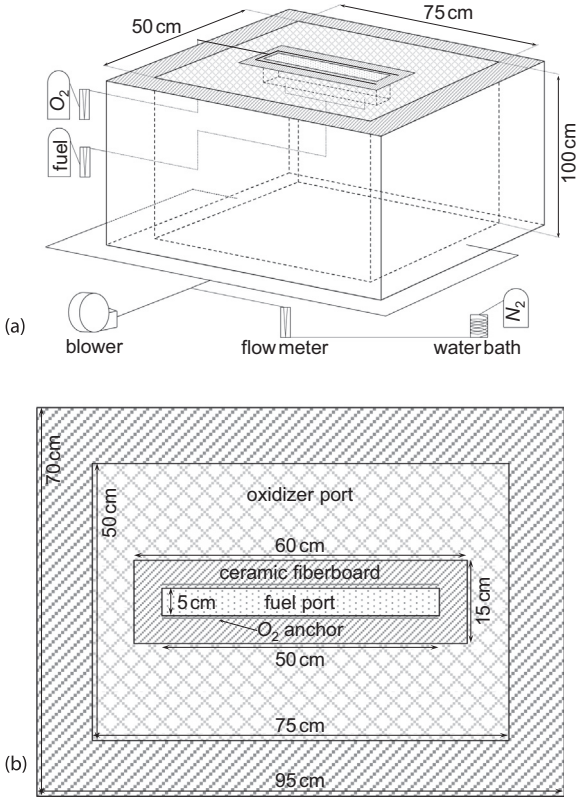

Fig. 1. (a) Diagram of experimental facility. (b) Top-view of burner/co-flow outlet. Reproduced from [9].