Available online at www.sciencedirect.com

ScienceDirect

Proceedings of the Combustion Institute xxx (2014) xxx–xxx

Proceedings of the Combustion Institute

www.elsevier.com/locate/proci

A computational and experimental study of coflow laminar methane/air diffusion flames: Effects of fuel dilution, inlet velocity, and gravity

S. Cao [a,*], B. Ma [a,1], B.A.V. Bennett [a], D. Giassi [a], D.P. Stocker [b], F. Takahashi [c], M.B. Long [a], M.D. Smooke [a]

[a] *Department of Mechanical Engineering & Materials Science, Yale University, P.O. Box 208284, New Haven, CT 06520-8284, USA*

[b] *NASA Glenn Research Center, Cleveland, OH 44135, USA*

[c] *National Center for Space Exploration Research on Fluids and Combustion, NASA Glenn Research Center, Cleveland, OH 44135, USA*

Abstract

The influences of fuel dilution, inlet velocity, and gravity on the shape and structure of laminar coflow CH_4–air diffusion flames were investigated computationally and experimentally. A series of nitrogen-diluted flames measured in the Structure and Liftoff in Combustion Experiment (SLICE) on board the International Space Station was assessed numerically under microgravity (μg) and normal gravity (1 g) conditions with CH_4 mole fraction ranging from 0.4 to 1.0 and average inlet velocity ranging from 23 to 90 cm/s. Computationally, the MC-Smooth vorticity–velocity formulation was employed to describe the reactive gaseous mixture, and soot evolution was modeled by sectional aerosol equations. The governing equations and boundary conditions were discretized on a two-dimensional computational domain by finite differences, and the resulting set of fully coupled, strongly nonlinear equations was solved simultaneously at all points using a damped, modified Newton's method. Experimentally, flame shape and soot temperature were determined by flame emission images recorded by a digital color camera. Very good agreement between computation and measurement was obtained, and the conclusions were as follows. (1) Buoyant and nonbuoyant luminous flame lengths are proportional to the mass flow rate of the fuel mixture; computed and measured nonbuoyant flames are noticeably longer than their 1 g counterparts; the effect of fuel dilution on flame shape (i.e., flame length and flame radius) is negligible when the flame shape is normalized by the methane flow rate. (2) Buoyancy-induced reduction of the flame radius through radially inward convection near the flame front is demonstrated. (3) Buoyant and nonbuoyant flame structure is mainly controlled by the fuel mass flow rate, and the effects from fuel dilution and inlet velocity are secondary.

Keywords: Laminar coflow diffusion flame; Gravity; Fuel dilution; Inlet fuel velocity; Flame shape and structure

* Corresponding author. Fax: +1 203 432 6775.
E-mail address: su.cao@yale.edu (S. Cao).
[1] Currently at GE Global Research.

http://dx.doi.org/10.1016/j.proci.2014.05.138
1540-7489/

Please cite this article in press as: S. Cao et al., *Proc. Combust. Inst.* (2014), http://dx.doi.org/10.1016/j.proci.2014.05.138

1. Introduction

Diffusion flames exist in most practical combustors, and an accurate understanding of their structure is crucial to efficiency improvement and pollution suppression. A coflow laminar diffusion flame, which has well-defined boundary conditions, is the simplest configuration from which interactions between flow field and reactions can be readily modified and studied [1]. Knowledge obtained from coflow laminar diffusion flames is not only of fundamental importance, but also can facilitate the study of turbulent diffusion flames in practical industrial combustors [2].

Microgravity provides an ideal environment for flame research. Compared to their counterparts in normal gravity, microgravity flames generally have larger temporal and spatial scales, they are exempt from intrusions of buoyancy forces, and they can be studied over a wider range of flame conditions. Over the past few decades, a significant amount of research has been conducted in microgravity, and the effects of various parameters on flame shape, structure, stabilization, and sooting behavior have been extensively studied (e.g., [3–11]).

In this work, previous computational and experimental investigations of coflow laminar diffusion flames (e.g., [8,9]) were further extended to characterize the effects of fuel dilution, inlet velocity, and gravity. The present work's objectives are: (1) to obtain an enhanced understanding of the influences of fuel dilution, inlet velocity, and gravity on the flame structure; and (2) to assess the accuracy of the numerical model by comparing computational and experimental results.

2. Burner configuration

The burner consists of a central jet, from which the fuel mixture issues, and a surrounding coaxial square duct, from which the coflow air flows. The inner jet's inner radius is $r_I = 0.162$ cm and its wall thickness is $w_{JET} = 0.028$ cm. The outer square duct's width is 7.62 cm. Details of the burner construction and operation are provided in a companion paper [12]. Eighteen flames have been studied under both microgravity (μg) and normal gravity (1 g) (36 flames total): three fuel dilution levels (40%, 70%, and 100% CH_4 in mole fractions) and six inlet velocities (approximately 24, 46, 55, 64, 80, 89 cm/s, changing slightly at different fuel dilutions). Since each flame is surrounded by an air coflow and takes at most 5% of the cross-sectional area of the burner, the square duct is approximated as a coaxial tube with an identical cross-sectional area (radius $r_O = 4.288$ cm); see Fig. 1 (left). The velocity profile of the fuel stream is parabolic, with the average Reynolds number in the fuel jet ranging from 15 to 87. Due to hardware imperfections, the flow field of the coflow air is not perfectly flat and has a bump near the inner tube; see Fig. 1 (right). To capture this nonideality, the measured inlet velocity distribution has been fitted as $v_z(r) = v_0[1 - \tanh((r - 0.67r_I)/0.038\text{ cm})] + 13.50[0.5 + 0.5\tanh((r - 1.1(r_I + w_{JET}))/0.09\text{ cm}) - \exp((r - r_O)/0.240\text{ cm})] + 3.16[\tanh((r - 0.44)/0.15\text{ cm}) + \tanh((0.96 - r)/0.35\text{ cm})]$, where the coefficient v_0 is determined numerically to match the inlet mass flow rate of the fuel mixture specified in the experiments. Since temperature measurements near the fuel tube exit are unavailable, the inlet temperature is set to 298 K.

3. Computational approach

The numerical framework is similar to those in the authors' previous works (see, for example, [13–16]) with the MC-Smooth vorticity–velocity formulation [17] employed. The gas is assumed Newtonian and diffusion is Fickian; the nth species diffusion velocity is calculated using detailed mixture averaging. The Soret and Dufour effects are neglected. The flow's small Mach number implies that the pressure field can be obtained via the ideal gas law. All thermodynamic, chemical, and transport properties are evaluated using vectorized and highly efficient libraries [18]. The gas-phase chemistry is the GRI 3.0 mechanism [19] with all nitrogen-containing species (except N_2) removed, leaving 35 species and 217 reactions. Certain reactions related to benzene and associated species (see Table 1 of [20]) were added to allow the simulation of the sooting process in the 70% CH_4 and 100% CH_4 flames, with the augmented mechanism containing 42 species and 250 reactions. Since the maximum soot volume fraction of all flames is 0.22 ppm, effects of radiation reabsorption are insignificant [15,16,21], and the power radiated from soot and gas bands (CO,

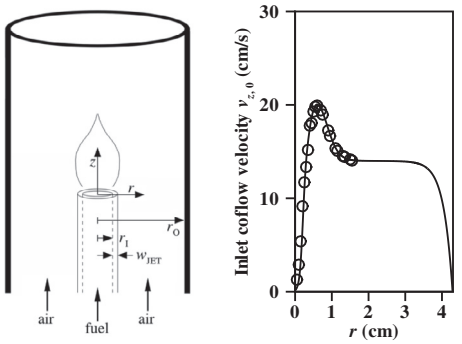

Fig. 1. Left: Schematic of the burner used in the simulations. Right: Measured (circles) and fitted (line) velocity profiles of the coflow air.

Please cite this article in press as: S. Cao et al., *Proc. Combust. Inst.* (2014), http://dx.doi.org/10.1016/j.proci.2014.05.138