Combustion and Flame 162 (2015) 2602–2609

Contents lists available at ScienceDirect

Combustion and Flame

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

Dynamics of bluff-body-stabilized premixed hydrogen/air flames in a narrow channel

Bok Jik Lee [a,*], Chun Sang Yoo [b], Hong G. Im [a]

[a] Clean Combustion Research Center, King Abdullah University of Science and Technology, Thuwal 23955-6900, Saudi Arabia
[b] School of Mechanical and Nuclear Engineering, Ulsan National Institute of Science and Technology, Ulsan 689-798, Republic of Korea

ARTICLE INFO

Article history:
Received 12 March 2015
Received in revised form 12 March 2015
Accepted 13 March 2015
Available online 4 April 2015

Keywords:
Bluff-body-stabilized flames
Premixed flames
Blow-off
Direct numerical simulation

ABSTRACT

Two-dimensional direct numerical simulations were conducted for bluff-body stabilized flames of a lean hydrogen/air mixture at near-blowoff conditions in a meso-scale channel. Parametric simulations were conducted by incrementally varying the inflow velocity in the vicinity of the blowoff limit, and the corresponding flame response was monitored. The present study is a showcase of combustion DNS with embedded boundary representation, and full demonstration of the detailed visualization of the near-blowoff flame characteristics. As the inflow velocity approaches blowoff limit, the flame dynamics exhibit a complex sequence of events, such as periodic local extinction and recovery, and regrowth of the bulk flame by the flame segments attached behind the bluff-body. The total extinction is observed as the attached flames shrink down and are no longer able to regrow the bulk flames. Despite the disparity in the physical scale under study, the observed sequence of the extinction pathway shows a strong similarity with experimental observations at larger scale combustion systems.

1. Introduction

Combustion in meso-/micro-scale dimension has been an active research subject for the past decades for its practical implication in the development of compact power generation devices [1,2]. From the fundamental combustion standpoint, one of the major challenges in developing micro-combustors is to achieve stable combustion in the presence of the increased heat losses associated with higher surface-to-volume ratio, which is an inherent nature of small scale devices. For premixed combustion systems, recent studies reported that adding a bluff-body stabilizer within a micro-channel can significantly extend the blowoff limits [3–6], in a manner similar to large scale gas turbine combustor applications. While a bluff-body has a desirable effect of flame stabilization by creating a recirculation zone, it may also pose adverse effects on flame stability by the unsteady vortex shedding into the flow field [7–9], causing fluctuations in the region of flame stabilization. Therefore, it is of fundamental and practical interest to understand the detailed physical mechanism of bluff-body flame stabilization within a narrow channel premixed combustor.

In an attempt to investigate the intricate flame stabilization dynamics, the existing experimental and computational studies of hydrogen/air flames in meso-scale combustors are limited in spatial and temporal resolution. For example, the pictures taken from the experiments by Wan et al. [3] only provided diffuse images of luminous reaction zones, while the steady numerical simulations could only reproduce statistically averaged flame positions. Such information is not sufficient to reveal all necessary details in order to understand the mechanism of the highly transient and localized flame stabilization and extinction events.

In the present study, high fidelity direct numerical simulations (DNS) are carried out with full temporal and spatial resolution in order to investigate the onset of instability of hydrogen/air premixed flames stabilized by a bluff-body in a meso-scale channel. The problem configuration considered is well suited for DNS in favor of its small physical dimension and the nearly two-dimensional flow characteristics. On the other hand, a unique challenge in computational implementation is to represent an immersed solid body in the computational domain within the high order finite difference framework. The present work showcases as one of the first attempts to employ an immersed body representation in combustion DNS applications.

Parametric simulations are conducted by sweeping through a range of inflow velocity as a main parameter, and the corresponding unsteady flow and combustion characteristics are investigated. The main scope of the present study is focused on the observation of the overall flame behavior through the analysis of a sequence of instantaneous solution fields for different inflow conditions, which unravels some new phenomena that have not been reported

* Corresponding author.
E-mail address: BokJik.Lee@kaust.edu.sa (B.J. Lee).

http://dx.doi.org/10.1016/j.combustflame.2015.03.009
0010-2180/

before. It is anticipated that the findings from such a meso-scale combustor will also provide valuable insights into the understanding of the bluff-body flame stabilization mechanism in macro-scale combustors, which is one of the most challenging practical questions in today's combustion research.

2. Configuration and numerical method

A two-dimensional computational domain (Fig. 1) is considered for a rectangular channel of height at $D = 1$ mm, and the length at $10D$. A bluff-body flame stabilizer is represented by a square block of size $0.5D$ by $0.5D$, whose center is located at $2.25D$ (i.e., the leading edge is at $2D$) downstream of the inflow boundary on the left.

Fully compressible, multi-species reactive Navier–Stokes equations are solved with a finite difference method using 8th order central difference and 4th order explicit Runge–Kutta time integration [10]. For a lean hydrogen/air mixture with the equivalence ratio of 0.5, a detailed reaction mechanism [11] with 9 species and 19 reactions is used. Chemical kinetics, thermodynamics, and transport properties are calculated using the subroutine modules adapting the Chemkin libraries [12,13]. Transport properties are computed based on the mixture-averaged formula. To fully resolve reaction layers, a Cartesian grid system using a uniform grid spacing of $\Delta x = 5$ μm is chosen after careful grid convergence tests. For parallel computing, a domain decomposition with MPI communications is employed with nearly linear scalability. A typical run with an Intel Xeon CPU cluster for time integration up to 10 ms takes approximately 6000 CPU hours.

For the inclusion of bluff-body geometry in the computational domain, an embedded boundary method is implemented. In this study, the square cylinder geometry is modeled as a union of logical MPI blocks, which is flagged to be excluded from the computational domain for time integration. The simplest way to implement the algorithm is to choose the boundaries of the object to be aligned with the grid lines, such that the object boundaries are treated as non-communicating physical boundaries in the same manner as the computational domain boundaries. All physical boundary conditions and one-sided derivative operations used at the domain boundaries are then directly applicable to the embedded boundaries.

No-slip velocity and adiabatic thermal boundary conditions are applied to the channel walls and bluff-body surfaces. For the small combustor dimension under study, the heat loss to the isolated small-scale bluff-body is negligible and the adiabatic condition for the immersed boundary is considered reasonable. Moreover, simulations were also conducted using a constant temperature channel wall condition at $T_{wall} = 298$ K, and no discernible differences were found in flame dynamics. Therefore, only the results with the adiabatic wall conditions are reported here. Non-reflecting characteristic boundary conditions are applied for the inflow/outflow boundaries to ensure no spurious acoustic wave reflections [14,15]. At the inflow boundary, a fully developed parabolic profile with a mean inflow velocity of U is imposed, with enforcing

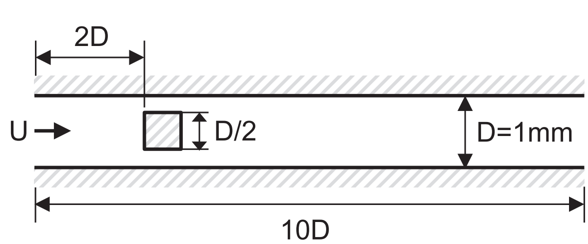

Fig. 1. Problem configuration with relative dimensions of a two-dimensional narrow channel with a square-cylinder flame stabilizer.

constant temperature $T = 298$ K, pressure $p = 1$ atm, and composition of fresh hydrogen/air mixture at the equivalence ratio of 0.5.

To investigate the dynamic responses of the flame stabilization and blow-off, the inflow velocity was the main parameter for the study. Cases with a mean inflow velocity U ranging from 15 m/s to 25 m/s are investigated. In each run, the inflow velocity is set at a constant value, and the subsequent temporal evolution of the flame dynamics is observed.

3. Non-reacting flows

As a reference case, non-reacting flow simulations are first conducted to examine the basic characteristics of the cold flows around the bluff-body in a meso-scale channel, for the same range of the inflow velocity to be considered in the reacting flow simulations.

Figure 2 shows instantaneous snapshots (at 10 ms into the simulation) of the vorticity isocontours for five different inflow velocity conditions. The flow residence time in the channel, i.e., the flow-through time, is 0.5 ms at $U = 20$ m/s. The corresponding flow Reynolds number based on the height of the bluff-body and the mean inflow velocity is 375, 450, 500, 550, and 625, respectively. The maximum level of the vorticity magnitude is set at 10^6/s. It is evident that the high fidelity simulations clearly capture the thin boundary layer development along the channel walls and the bluff-body, which subsequently leads to interactions of vortices originating from the two boundary layers further downstream. The separation zones formed on top and bottom sides of the bluff-body are repeatedly opened and closed at the trailing edge due to the interaction of the flow with the recirculation zone at the base of the bluff-body. The flows behind the bluff-body are highly unsteady and asymmetric, and the vortex-shedding becomes stronger and more asymmetric as the mean inflow velocity is increased.

The frequency f of asymmetric shedding of the vortices is found to be approximately 24 kHz for the non-reacting cases at $U = 20$ m/s, and the corresponding Strouhal number, defined as $\mathrm{St} = fD/2U$, is about 0.6. The velocity within the narrow path between the bluff-body and the channel wall is much larger than the mean inflow velocity, measured to be around 60 m/s, due to the significant contraction of flow passage both by the blockage ratio of 0.5 and by the boundary layer development. If this local velocity is used, the Strouhal number becomes approximately 0.2, which is close to the values commonly found in unconfined flows around bluff-bodies.

4. Reacting flows

Following the nonreacting flow simulations, fully reacting calculations are conducted for the same range of inflow velocities. Flame dynamics observed along the inflow velocity approaching blowoff limit are described in this section. Simulations are conducted up to $t = 5$ ms (ten flow-through times) or longer until the transient characteristics are settled and a limit cycle behavior is reached. Starting from a fresh simulation at $U = 15$ m/s with an ignition source, successive simulations at incrementally higher inflow velocities are restarted from each of the previous solutions, resetting the simulation time to be zero upon restarting. At restart, a time-dependent inflow boundary condition is imposed with a smooth ramp-up of the mean inflow velocity from previous to new condition for a duration of 0.5 ms, which corresponds to approximately one flow-through time, in order to minimize additional transient effect due to the inflow acceleration. For the inflow velocity greater than 19 m/s, the flames exhibit unsteady limit-cycle behavior. For these simulations, the solution for $U = 19$ m/s at $t = 10$ ms is always used as the initial condition for the restart and the inflow velocity ramp-up was imposed subsequently, in