Combustion and Flame 174 (2016) 152–165

Contents lists available at ScienceDirect

Combustion and Flame

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

Turbulence/chemistry interactions in a ramp-stabilized supersonic hydrogen–air diffusion flame

Jesse A. Fulton[a], Jack R. Edwards[b,*], Andrew Cutler[c], Jim McDaniel[d], Christopher Goyne[d]

[a] Sandia National Laboratories, Albuquerque, NM 87114, United States
[b] North Carolina State University, Mechanical and Aerospace Engineering, Raleigh, NC 27695, United States
[c] The George Washington University, Newport News, VA 23602, United States
[d] University of Virginia, Mechanical and Aerospace Engineering, Charlottesville, VA 22904, United States

ARTICLE INFO

Article history:
Received 8 February 2016
Revised 6 April 2016
Accepted 15 September 2016

Keywords:
Supersonic combustion
Large-eddy simulation

ABSTRACT

Hybrid large-eddy / Reynolds-averaged Navier–Stokes simulations of turbulence / chemistry interactions occurring within a ramp-injected, hydrogen-fueled scramjet combustor are presented in this work. The experimental geometry is one of several studied at the Universty of Virginia as part of the National Center for Hypersonic Combined Cycle Propulsion and consists of an isolator, a combustor, and an extender section. Data collected includes coherent anti-Stokes Raman spectroscopy (CARS) measurements of major species composition and temperature at several streamwise planes, stereoscopic particle image velocimetry (PIV) measurements, hydroxyl planar-induced fluorescence (OH-PLIF) imagery, wall pressure distributions, and line-of-sight profiles of temperature and water concentration obtained using tunable diode laser spectroscopy (TDLAS). This paper focuses on an equivalence ratio of 0.17, which does not produce enough heat release to force a shock train into the isolator. The computational methods utilize a hybrid fourth-order central-difference / upwind strategy to enable accurate resolution of turbulent structures and employ a nine-species hydrogen oxidation mechanism. Generally accurate predictions of temperature, velocity, and nitrogen mole fraction are achieved through a 'laminar chemistry' assumption for the filtered species production rates, though results do improve slightly with the use of a simple turbulence / chemistry subgrid closure model. The predictions are most sensitive to the choice of isolator inflow boundary condition, with the use of a recycling / rescaling technique to sustain turbulent fluctuations resulting in an 'over-mixing' effect immediately downstream of the fuel injector. Turbulence–chemistry interactions in the flameholding region are examined from the standpoint of laminar flamelet theory. A region of high scalar dissipation rate, coincident with the breakdown of the fuel plume and the interaction of a reflected shock wave with the plume, inhibits flame propagation, forming a 'hole' in the flame. Advection of cooler fluid downstream into regions of moderate scalar dissipation enlarges the 'hole', but eventually the flame reconnects. These results point to one potential disadvantage of fuel-air mixing technologies that enhance axial vorticity – even if conditions for combustion are favorable, high strain rates generated by the interaction and breakdown of vortex pairs can lead to flame suppression.

1. Introduction

The path to extend airbreathing flight to hypersonic speeds has been accompanied by many technical challenges. As combustor flow speeds become supersonic, the flow residence time in the combustor decreases; as a consequence, there is now the problem of maintaining mixing and reaction rates high enough to permit complete combustion before the flow exits the combustor. The Reynolds number of the engine fluid flow also increases with Mach number, leading to thick turbulent boundary layers which may interact with shock waves and may respond to heat release generated through combustion. In order for airbreathing hypersonic propulsion systems to be successfully designed and operated, a thorough understanding of the complex reacting flow in ramjet and scramjet combustors is necessary. Achieving this knowledge through experimentation alone is accompanied by many difficulties, such as test-time limitations, optical access limitations, and the inability of ground-based experiments to properly match flight conditions.

These issues in hypersonic research have been alleviated to some extent with the ongoing maturation of numerical simulation technology. However, there are still aspects of a high-speed

---

* Corresponding author. Fax: +1 919 515 7968.
E-mail addresses: jredward@eos.ncsu.edu, jredward@ncsu.edu (J.R. Edwards).

http://dx.doi.org/10.1016/j.combustflame.2016.09.017
0010-2180/

propulsion system's operation which lack much understanding. Of these, one which requires perhaps the most attention is the interaction between turbulence, mixing, and reactive kinetics within a scramjet combustor. Large-eddy simulation (LES) and hybrid LES/RANS simulation methods offer several advantages over state-of-the-practice Reynolds-Averaged Navier–Stokes (RANS) methods in capturing this interaction, as they are able to directly resolve large-scale turbulent structures and their corresponding effects. However, since molecular mixing and combustion take place at the unresolved subgrid scales, there is still a need to model the effects of reactivity as observed at the resolved scales.

Several recent studies have utilized LES or LES/RANS techniques for modeling scramjet combustion. A combustor mapped at DLR [1] has been studied by Berglund and Fureby [2], Potturi and Edwards [3], and Genin and Menon [4]. The results showed variations in the prediction of shear layer spreading rate and peak temperatures depending on the mesh resolution, the assumed degree of three-dimensionality, and the subgrid and combustion models employed. A combustor tested at Japan's National Aerospace Laboratory has been the subject of recent studies using LES methods by Berglund, et al. [5], Fedina et al. [6], and Sabelnikov, et al. [7] These simulations studied two strut injector orientations. The results provided insights into the combustion behavior and produced combustor wall-pressure data which agreed well with experimental measurements [8].

More recent efforts using partially-stirred reactor (PaSR) [9] and flamelet-type [10] combustion models have focused on the hydrogen-fueled HyShot combustor, ground-tested at DLR. Good agreement with available wall-heating and wall-pressure data has been generally observed, irrespective of the combustion model employed. Large-eddy simulations have also been performed by Koo et al. [11] and Fureby et al. [12] for a hydrogen-fueled combustor tested at the University of Michigan [13]. These studies utilized very different turbulence–chemistry interaction models but were able to predict the observed transition between cavity-stabilized and jet-wake stabilized modes of operation.

Some LES and hybrid LES/RANS studies of hydrocarbon-fueled scramjet combustion have also been performed, with simulations of the HiFire combustor using a flamelet / progress-variable method [14] and simulations of cavity-stabilized ethylene combustion experiments performed at the University of Virginia (UVa) [15] being two recent examples. Comparisons with observed wall pressures and (in the case of the University of Virginia experiments) tunable diode laser spectroscopy (TDLAS) measurements have revealed adequate agreement with experimental observations.

A common thread, however, among all of the studies previously mentioned, is the relative paucity of experimental data upon which to base a substantive assessment of the interplay among the various factors that might affect the outcome of a large-eddy simulation. In addition to the subgrid-closure issues previously mentioned, these factors may include facility-dependent effects, such as variable wall heating loads and inflow non-uniformities, as well as numerical effects, such as the choice of spatial discretization and the mesh resolution. A clearer evaluation of the potential of large-eddy simulations for high-speed combustion requires more detailed experimental data.

The National Center for Hypersonic Combined-Cycle Propulsion (NCHCCP) was a program developed to provide a mechanism for intense collaboration between numerical and experimental groups in order to gain a deeper understanding of hypersonic propulsion. One of the primary goals in this center was to provide benchmark-quality datasets for scram- and dual-mode hydrogen combustion using the Supersonic Combustion Facility (SCF) located at the University of Virginia. Previous studies focusing on the 'Configuration A' combustor geometry without an isolator have been described in [16–19] while companion numerical predictions using a hybrid LES/RANS method have been reported in [20,21]. The general outcome of these studies was that even simple closures, such as the 'laminar chemistry' assumption often invoked for high-speed combustion, were able to provide adequate agreement with mean-flow experimental data (determined from Coherent Anti-Stokes Raman Spectroscopy (CARS), Stereoscopic Particle Image Velocimetry (SPIV), Wave Modulation Spectroscopy (WMS), and OH-Planar Induced Fluorescence (OH-PLIF) measurements, among other diagnostics). The influence of the choice of reaction mechanisms on the obtained predictions was stronger that any tested variation in subgrid combustion model (though only relatively simple closures were assessed). [21]. A disadvantage of the Configuration A studies is that the spatial resolution of the CARS measurements was not as desired due to problems in the stepper motor used to position the lasers. A further disadvantange is the absence of an isolator, which restricted the equivalence ratio to low values (<0.3). These issues were addressed through the construction of a new rig, termed 'Configuration C', and the development of detailed maps of the flow field using the diagnostics mentioned above at conditions that yield 'scram mode' operation ($\Phi = 0.17$; no shock train in the isolator) and 'dual-mode' operation ($\Phi = 0.49$; shock train in the isolator).

This paper describes some of the computational studies performed for the Configuration C experiments at 'scram-mode' conditions, with one objective being to evaluate aspects of the computational model through quantitative comparisons with some of the available experimental data. Another objective is to utilize the computational results, processed according to concepts from flamelet theory, to examine aspects of flame stabilization in the near field of the fuel injector.

2. Facility

The University of Virginia's Supersonic Combustion Facility (SCF) is a small-scale direct-connect dual-mode experimental scramjet combustor apparatus whose operation is designed to simulate flight conditions at Mach 5 flight enthalpy – see Figs. 1 and 2. It is vertically mounted and consists of an inlet nozzle, constant-area isolator, combustor, and extender, from which the exhaust gases are vented to the atmosphere. Figure 1 shows an older flow-path configuration; Fig. 2 gives detailed dimensions of Configuration C, which was the actual configuration used in this study.

The Mach 2 inlet nozzle, located upstream of the isolator and combustor apparatus, is responsible for bringing the incoming airflow up to the speeds at the isolator entrance that would normally be seen during low hypersonic flight, after the freestream air was slowed and compressed in the inlet section of a hypersonic aircraft. In addition, the incoming air is compressed to a stagnation pressure of roughly 300 kPa using an oil-free compressor, and a 300 kW, 14-stage electrical resistance heater raises the stagnation temperature to 1200 K without introducing flow vitiates – a significant advantage of this facility. The facility is capable of operating continuously for long durations, limited only by the available fuel supply during combusting experiments.

For the work presented here, hydrogen is used as the fuel for the SCF. The fuel is introduced into the airflow via a single, wall-mounted, unswept, 10-degree raised compression ramp injector. The height of the injector at its end plane is 0.635 cm and its width is 1.27 cm. The fuel-injection nozzle is a converging-diverging conical design that accelerates the fuel to approximately Mach 1.7 at its exit. At lower equivalence ratios, the facility operates in scramjet mode; as the equivalence ratio is increased, it transitions to dual-mode operation.

Operational data for the SCF, as well as some other parameters, are summarized in Table 1.