



Effect of flow distortion on fuel/air mixing and combustion in an upstream-fueled cavity flameholder for a supersonic combustor



Steven Etheridge^a, Jong Guen Lee^{a,*}, Campbell Carter^b, Mark Hagenmaier^b, Ryan Milligan^c

^a University of Cincinnati, Cincinnati, OH 45221, United States

^b U.S. Air Force Laboratory, Wright-Patterson Air Force Base, OH 45433, United States

^c Taitech, Inc., Beavercreek, OH 45430, United States

ARTICLE INFO

Article history:

Received 23 February 2017

Received in revised form 14 June 2017

Accepted 20 June 2017

Available online 23 June 2017

Keywords:

Supersonic combustor

Shock wave interaction with flow

Fuel air mixing

Flame stabilization

ABSTRACT

This paper describes an experimental study of the effects of an incident shockwave on the flow field, fuel distribution and combustion within a cavity flameholder with upstream fuel injection. Two impingement locations are employed: (1) near the fuel injector (the so-called *shock-on-jet* case) and (2) on the cavity shear layer (the *shock-on-cavity* case). Shadowgraph is used to characterize the flow field. Air seeded with nitric oxide (NO) is used as the simulated fuel and the resulting planar laser-induced fluorescence (NO-PLIF) from NO molecules is used to characterize fuel/air mixing while planar laser-induced fluorescence of OH molecules to characterize the actual combustion process. The shadowgraph and NO-PLIF images are compared with a CFD (Computational Fluid Dynamics) solution of the Reynolds-averaged-Navier Stokes (RANS) for assessment and explanation of experimental results of non-reacting tests. The effect of the shock on the cavity shear layer is to control the fuel distribution within the cavity. The effect of the shock on the jet is to force the shear layer deep within the cavity, which results in higher fuel concentrations near the cavity centerline. The *shock-on-cavity* case causes the shear layer to separate upstream of the cavity. Mixing uniformity is enhanced by the increased breakup of the fuel plume. Combustion is stronger and more uniform with the shock impinging on the cavity, while it is limited to the edges of the cavity with shock impingement on the jet. The greater mixing afforded in the *shock-on-cavity* case reduces the fuel concentration near the centerline and allows stronger burning in the center of the cavity. Doubling the fuel injection momentum flux ratio does not strongly affect the pattern of fuel distribution in either case, but combustion in the *shock-on-cavity* case is reduced, because the fuel concentration at the centerline is high.

© 2017 Elsevier Inc. All rights reserved.

1. Introduction

High-speed air-breathing propulsion systems present unique difficulties in combustor design. At hypersonic flight speeds, it becomes necessary to maintain the supersonic velocities of the incoming airstream to prevent unacceptable total pressure losses and endothermic dissociation reactions that reduce the efficiency of the system. Supersonic velocities within the combustor reduce the airstream residence time to the order of 1 ms, requiring very efficient and rapid fuel injection, mixing and vaporization before combustion. Further, airstream speeds on the order of 1 km/s outpace the maximum turbulent flame speed (of order 10 m/s), rendering typical flame-holding methods impractical. Additionally,

combustor designs must account for changes in the flowfield experienced during acceleration and maneuvers that may affect fuel injection and flame-holding schemes. One common scheme is a cavity flame holder with upstream fuel injection from the combustor wall.

Numerous studies on gaseous fluid injection into a supersonic crossflow and the combustion performance of cavities have been conducted. Baurle and Gruber [1] used CFD to examine the effects of cavity length, depth and aft-wall angle on residence time and entrainment rate. Their results indicate that cavity geometry has no strong effect but residence time is influenced by cavity depth and entrainment rates are affected by cavity length. Hsu et al. [2] used spontaneous Raman scattering to evaluate the fuel distribution of a passively fueled cavity in a Mach 2 flow and found that fuel entrainment is highly dependent on the location of the shear layer and the interaction with the cavity aft-wall. Jet penetration also affects fuel entrainment, with higher jet injection momentum

* Corresponding author at: Department of Aerospace Engineering and Engineering Mechanics, Engineering Research Center, Room 484, PO Box 210070, Cincinnati, OH 45221-0070, United States.

E-mail address: jongguen.lee@uc.edu (J.G. Lee).

flux ratios resulting in greater penetration but lower entrainment rates, as less fuel is captured by the shear layer. The effect of high backpressure—simulating the pressure rise from cavity and main flow combustion—causes the shear layer to separate and lift above the cavity, further reducing fuel entrainment rates. Hsu et al. suggested that direct cavity fueling may reduce these dependencies. Later, Gruber et al. [3] used NO-PLIF (planar laser-induced fluorescence of the NO molecule) to study the effect of fueling location on fuel distribution within the cavity. Normal and angled upstream injection each provided low levels of entrainment within the cavity; two other direct fueling configurations were also tested. Injection just downstream of the rearward facing step led to improved fuel distribution but was still dependent on the shear layer interaction with the ramp. Direct injection in the upstream direction from the aft-ramp provided the best fuel distribution within the cavity. Gruber et al. [4] used shadowgraph and schlieren imaging as well as CFD to study drag coefficients, residence times and pressure fields for various cavity geometries, concluding that cavities with aft-wall angles less than 90° result in more stable, 2D flows without the high pressure oscillations experienced in cavities with 90° wall angle. Sun et al. [5] used acetone PLIF and CFD to examine the fuel distribution of an upstream fuel jet around a cavity. They found that most of the fuel from the jet is entrained by counter-rotating vortices and lifts away from the combustor wall but also that the vortex pair induces vortices within the cavity shear layer that captures some of the fuel from the jet; in this manner, some fuel is then entrained into the cavity.

In most studies of supersonic combustion, the flow entering the test section first passes through a nozzle designed to make the flow as uniform as possible. This is in contrast to the flow produced by flight vehicle inlet, where the flow entering the combustor will have inherent distortions arising from the inlet and isolator geometry; furthermore, the nature of these distortions such as the strength and angle of these shocks will vary according to the flight envelope. A few studies have attempted to incorporate the effect of shockwaves within the combustor on fuel injection and flame holding. Gruber and Hagenmaier [6] developed a methodology for the design of a distortion generator using CFD and experiment. With their distortion-generator design, they attempted to eliminate the abrupt expansion caused by the back face of a compression wedge sometimes used to simulate inlet distortion effects. Ryan et al. [7] later used the generator developed by Gruber and Hagenmaier in an OH-PLIF study of a cavity with up- and downstream fueling; however, they did not specifically examine the effect of the flow distortion. Mai et al. [8], employing particle image velocimetry (PIV), found that the recirculation zone created behind the fuel jet, which was largely empty of fuel when there was no shockwave, was enlarged due to the presence of the shock and that the region of negative streamwise velocity was increased in volume. Additionally, with the shock impinging downstream of the jet, Mai et al. found that the fuel concentration within the recirculation zone was markedly increased. Their CFD results showed that these two effects led to an increase in the size of the combustible region behind the jet. However, when the shock impinges upstream of the injection point, Mai et al. observed that combustion could not be sustained. Campioli [9] and Schetz [10] studied the effect of the generated vorticity on the mixing process of a transverse fuel jet into a supersonic flow. Their results show that when the shock position is upstream of the injection point, the plume height is reduced because the static pressure of the main flow is higher behind the shockwave. The presence of the shock impinging downstream of the injection point was found to have a few key effects: (1) the plume penetration into the main flow is reduced by the flow turning towards the wall; (2) mixing is improved by the increase in vorticity previously discussed; (3) mixing is also improved as the shockwave impinges closer to the

injection location, due to the large gradient created by the shock residing within the plume before it begins to dissipate.

The objective of the present study is to examine the effect of an incident shockwave on (1) the flowfield of a single fuel jet upstream of a cavity, (2) the resulting fuel distribution and (3) combustion in the cavity; a companion study [11] was also conducted for the nonreacting flowfield using PIV. That work documents the significant changes in the cavity flowfield brought on by the distortion generator for both locations (and a third location not included in this study). In this work, two shock angles were studied as well as two shock impingement locations, with varying fuel injection momentum ratios. Shadowgraphy was used to image the flowfield, NO-PLIF was used to obtain the fuel/air mixing distribution in the cavity and OH-PLIF was used to characterize the intensity of combustion in the cavity. Experimental results obtained with one shock angle were compared with CFD solutions to gain physical insight on the effect of shockwave on the flow field and fuel/air mixing.

2. Experimental apparatus and numerical approach

2.1. Facility

Experiments are conducted in the supersonic wind-tunnel facility (Research Cell 19) [12,13] located at the Air Force Research Laboratory at Wright-Patterson Air Force Base. The facility provides a continuous source of clean compressed air at stagnation conditions up to 922 K (1660 R) and 5.27 MPa (764 psi) at a flow rate up to 15.4 kg/s (34 lbs/s). Further facility descriptions can be found elsewhere [12,13]. The airstream is accelerated to Mach 3 by a set of asymmetric half-nozzles. Visualization of flow field and measurement of fuel-air mixing is taken under “cold flow” conditions to simplify measurements at the stagnation temperature of 294 K (529 R) and the stagnation pressure of 1.2 MPa (175 psi). Combustion tests are conducted with heating the air (employing a heat exchanger) to approximately 644 K (1159 R) at the same stagnation pressure as the cold flow tests. Ethylene is used in combustion tests as the fuel.

Fig. 1 shows a schematic of the test section. Flow proceeds from left to right. The center of the upstream injector provides the origin for the cavity coordinate system, with the flow direction (streamwise) in the positive x-axis, the vertical direction provides the positive y-axis (transverse) and the width of the test section serves as the z-axis (spanwise). The constant area test section (5.08 cm high by 15.24 cm wide) is followed by a 2.5° divergence ramp on the bottom wall. The cavity flameholder, located on the bottom wall, spans the entire width of the tunnel test section; the cavity close-out is formed by a ramp at a 22.5° angle. The cavity is fueled from the ramp face with an array of 11 injectors (2 mm in diameter) that are spaced 12.7-mm (0.5 in) apart. The cavity is 16.5-mm

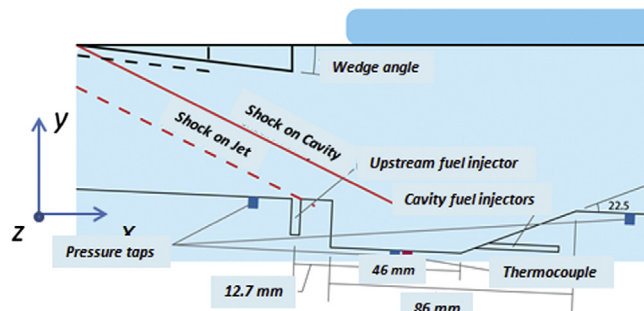


Fig. 1. Cavity schematic of the tunnel centerline, blue shaded regions indicate side and top window locations.

Download English Version:

<https://daneshyari.com/en/article/4992599>

Download Persian Version:

<https://daneshyari.com/article/4992599>

[Daneshyari.com](https://daneshyari.com)