Fuel Injection and Combustion Study for Mach 10-12 Scramjet

Dora E. Musielak
University of Texas at Arlington
dmusielak@uta.edu

ABSTRACT

A research program to support the development of scramjet engine with an axisymmetric geometry for a hydrogen-fueled vehicle capable of operation in the range of flight Mach numbers between 10 and 12. Using NASA's VULCAN code, a comprehensive CFD study was carried out to simulate the high speed reacting flows inside the combustion chamber. A 7 species/7 reaction hydrogen air kinetic model was used as baseline to study ignition and flame-holding of several fuel injection configurations. This paper reports results of jet penetration, mixing, and chemical reaction performance of a 3-D combustor with multiple wall-mounted fuel injectors arranged in two distinct configurations.

Key words: *scramjet, staggered fuel injection, mixing, chemical reaction, high-speed air breathing propulsion*

I. INTRODUCTION

Development of efficient supersonic combustion ramjets (scramjets) is crucial for realizing practical high-Mach hypersonic high speed flight within the atmosphere. The most fundamental issues in scramjet propulsion research iinclude flow path development, which in tune requires a deep understanding of the physical processes that underlie fuel injection, fuel-air mixing and combustion in the high-velocity, high-enthalpy flow environment. One of the main technology problems still unresolved concerns the effective fuel injection since the flow has very short combustor residence times, compressibility effects tend to cause poor mixing, and reaction must occur almost instantaneously. There are inherent losses in the supersonic incoming airstream. Thus, additional losses due to fuel injection, mixing, and combustion in the form of shock wave losses and pressure and friction drag, shear layer mixing losses, and loss of momentum of fuel jets therefore must be kept to a minimum and, at the same time, the most complete fuel-air mixing and fuel chemical release must be achieved to maximize thrust.[1]

For operation in the regime of flight Mach numbers between 10 and 12, the airflow in the scramjet combustor will be roughly between 3.3 and 4.7 at the entrance, depending on vehicle inlet design. If the length of the combustor is to be kept to a minimum, the air and injected fuel must mix adequately and combust in a time period no longer than a few thousandths of a second. A reduced degree of fuel-air mixing leads to an overall decrease in combustion efficiency and thrust.[2] For the past two decades most of the research effort in hypersonics has focused on developing fuel injection techniques to maximize mixing in order to reach the fuel-air ratios required for complete combustion.[3,4,5,6,7,8]

The flow within the scramjet engine is rather complex. The shock waves developed at the inlet and associated thick boundary layer will inevitably extend to the combustion region. This will result in shock-boundary layer interaction inside the combustor, which in turn may induce boundary layer separation. Furthermore, nonparallel fuel injection creates a bow shock ahead of the injector and may cause the flow to separate. The bow shock also increases the local temperature and pressure. These complex flow processes must be managed correctly.

The National Aerospace Initiative (NAI), the Program Executive Office Missiles and Space (PEO MS), and the Aviation and Missile, Research, Development and Engineering Center (AMRDEC) undertook efforts to develop air-breathing scramjet interceptors that are fueled by hydrogen. Hydrogen remains as the only fuel of choice for high velocity scramjets at flight Mach numbers greater than Mach 8 because of the necessity for rapid mixing and fast chemical reactions. However, although in theory hydrogen fueled scramjets appear simple, supersonic reaction of H_2 in air (O_2) remains a complex problem. Chemical kinetics, temperature, pressure, equivalence ratio, mixing rate and stream velocity all affect the rate of reaction. The ignition delay time of hydrogen-air mixtures is a limiting factor. To ensure ignition and stable burning, various combustion enhancement schemes have been proposed such as the usage of silane (SiH_4) to initiate or sustain H_2 combustion.

The reaction rates which control chemical reaction of fuel and air mixtures must be very fast. The main requirement is to minimize the induction time or ignition delay time. At high Mach number flight, static temperature and pressure conditions are typically high enough to autoignite fuels like hydrogen. Autoignition of hydrogen is easier in certain injector configurations such as from a strut because of the smaller boundary layer and higher surface temperature of the struts. However, struts are not effective or practical in all applications.

Also, flameholding devices and silane pilots are required in small-scale models to increase hydrogen reaction performance. For example, combustion characterization studies with hydrogen-air mixtures at high Mach conditions revealed a lack of sufficient mixing resulting in low combustion efficiency.[9] The gas temperature is also a contributing factor. If the temperature at the combustor entrance is too high the temperature of the combustion gas may exceed 2500 K and energy losses due to thermal dissociation of the gas and heat loss to the wall become large. The possibility of improving the performance of a Mach 12 scramjet engine by optimized control of the gas temperature in the combustorhas been investigated.[10]

Motivated by those challenges, we have studied numerous kinds of mixing enhancement techniques for compressible planar shear layers, and we considered many fuel injection methods.[11] Upstream fuel injection is another area of investigation that has been considered in the past 30 years and continues to be pursued. Meanwhile, introduction of nano-sized stimulant particles in the fuel to increase burning rate has been proposed but the results obtained to date are not conclusive.[12]

One of the most challenging supersonic combustion problems is that of mixing. If fuel is not properly injected and mixed into the supersonic airstream, it will not ignite, regardless of the pressure, temperature, or equivalence ratio. Due to compressibility effects, fuel injection is very difficult. The air stream is at such high pressure and velocity that fuel jet has a tendency to be pushed against the wall and injection is rendered ineffective.

To initiate the Scramjet Development Program, it was decided to study several aspects of scramjet operation. To address the issues addressed above, it was decided to use computational fluid dynamics (CFD) tools to simulate the injection of hydrogen, mixing with the oxygen in the airstream, and the combustion process and select the best fuel injection approach for application to a Mach 10-12 vehicle. This research study was envisioned to play an essential role in developing high Mach hydrogen-fueled hypersonic vehicles. The research is also important in help us understand and scope the experimental program that will be part of the development work for future hypersonic wind tunnels.

2. RESEARCH OBJECTIVES

The overall goals of the Scramjet Technology Development Program are to establish a new R&D program that would complement and support the future hypersonic wind tunnel. These goals include:

- Design a scramjet inlet-to-nozzle flow path for Mach 8-12 vehicles.
- Develop fuel injection techniques to maximize combustion efficiency.
- Guide the hypersonic wind tunnel experiments and scramjet engine testing.

To achieve those goals, a series of technical objectives were formulated. One objective was to conduct a numerical investigation of fuel injection techniques that promote mixing and chemical reaction for application to Mach 10-12 vehicles. The study included computational modeling and CFD simulation of hydrogen injection, mixing, ignition, and chemical reaction to identify fuel techniques that enhance mixing and combustion efficiency in a hydrogen-fueled scramjet engine. Specifically, the study focused on developing a fuel injection scheme that combines inlet injection with multiple injectors in the combustion chamber and the effect of cavity flameholders.

This research program was envisioned to support efforts that focus on development of scramjet engine concepts for a Mach 10-12 hydrogen-fueled vehicle, as reported in (11). For this study, a generic scramjet engine concept was adopted for the study, as shown in Fig. 1. It has an axisymmetric geometry and would operate in the range of flight Mach numbers between 10 and 12.

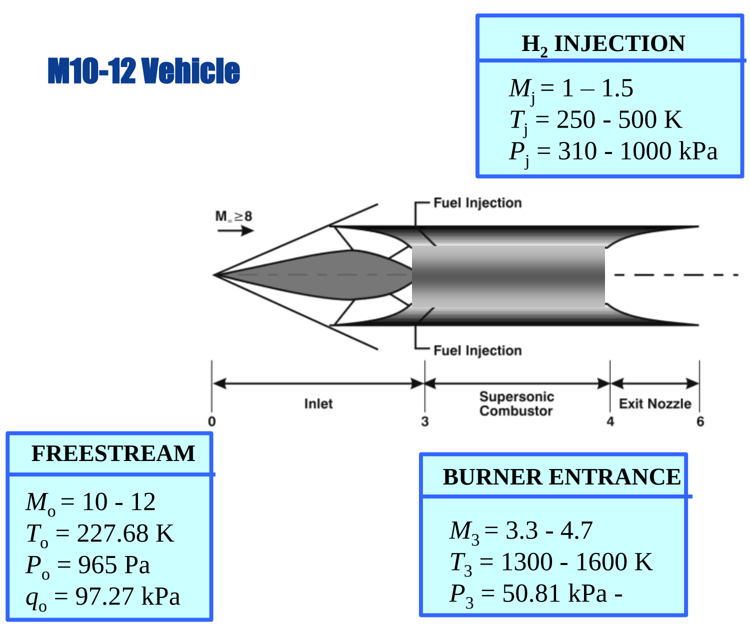


Figure 1. Conceptual Mach 10-12 Scramjet Engine.

Ignition and flame-holding are two important factors that have to be addressed in the design of a scramjet's fuel injection system.[13,14] The primary objective of a flame holder in supersonic combustion is to reduce the ignition delay time and provide a continuous source of radicals for the chemical reaction to be established in the shortest distance possible. In general, flame holding is achieved by three methods: (1) organization of a recirculation area where the fuel and air can be mixed partially at low velocities, (2) interaction of a shock wave with partially or fully mixed fuel and oxidizer, and (3) formation of coherent structures containing unmixed fuel and air, wherein a diffusion flame occurs as the gases are convected downstream. These stabilization techniques were studied as part of this work,[15] and are summarized below.

The simplest flame stabilization approach was implemented with transverse (normal) injection of hydrogen fuel from a wall orifice, and some degree of flame-holding was achieved. It was observed that as the fuel jet interacts with the supersonic crossflow of air, a bow shock is produced (see Fig. 3). As a result, the upstream wall boundary layer separates, providing a region where the boundary layer and jet gas mix subsonically upstream of the jet exit plane. This flow region is very important in transverse fuel injection designs because of its flame-holding capability. However, this injection strategy has stagnation pressure losses due to the strong 3-D bow shock formed by the normal jet penetration at the high flow velocities. Of course, it is possible to reduce the pressure losses by injecting the fuel at an angle, as it was done using 30° injection.[15] In this case, the resulting bow shock is weaker; and although it is believed that the jet axial momentum can also contribute to the net engine thrust, it was found in this study that, with the 30° jet angle, the smaller recirculation region did not significantly improve the mixing and combustion efficiency.

Furthermore, since the component of a scramjet propulsion system that determines its overall efficiency is the vehicle inlet, it was decided to add an analysis of inlet flows. At high flight speeds, vehicle inlet requirements change markedly, since the optimal inlet shape becomes dependent on the engine's operating conditions, i.e., on the pressure, temperature and on the excess fuel/air ratio (stoichiometric or equivalence ratio). A change in the inlet shape entails both a change in the combustor geometry and in the level of total pressure loss in the combustor, for the Mach 10-12 scramjet concept under consideration. Another way of achieving flame stabilization is by means of a wall cavity. Results of such study are reported elsewhere.[16]

3. STUDY METHODOLOGY AND PHYSICAL MODELS

To identify fuel injection techniques that enhance mixing and improve combustion in a scramjet engine, a computational fluid dynamic (CFD) simulation approach was selected, as it is the most readily available tool to obtain a reasonably prediction of those processes. Using CFD analysis can also serve as a guide for defining the experimental test program for the future hypersonic wind tunnel.

Figure 2 illustrates the physical model used as a baseline for the fuel injection studies. As shown, a slice of the annular scramjet combustion chamber is taken and assumed to be almost rectangular to represent the 3-D combustor model for the CFD analysis. The computational domain is 30.5 cm long, with a constant cross section of 7.62 × 7.0 cm.

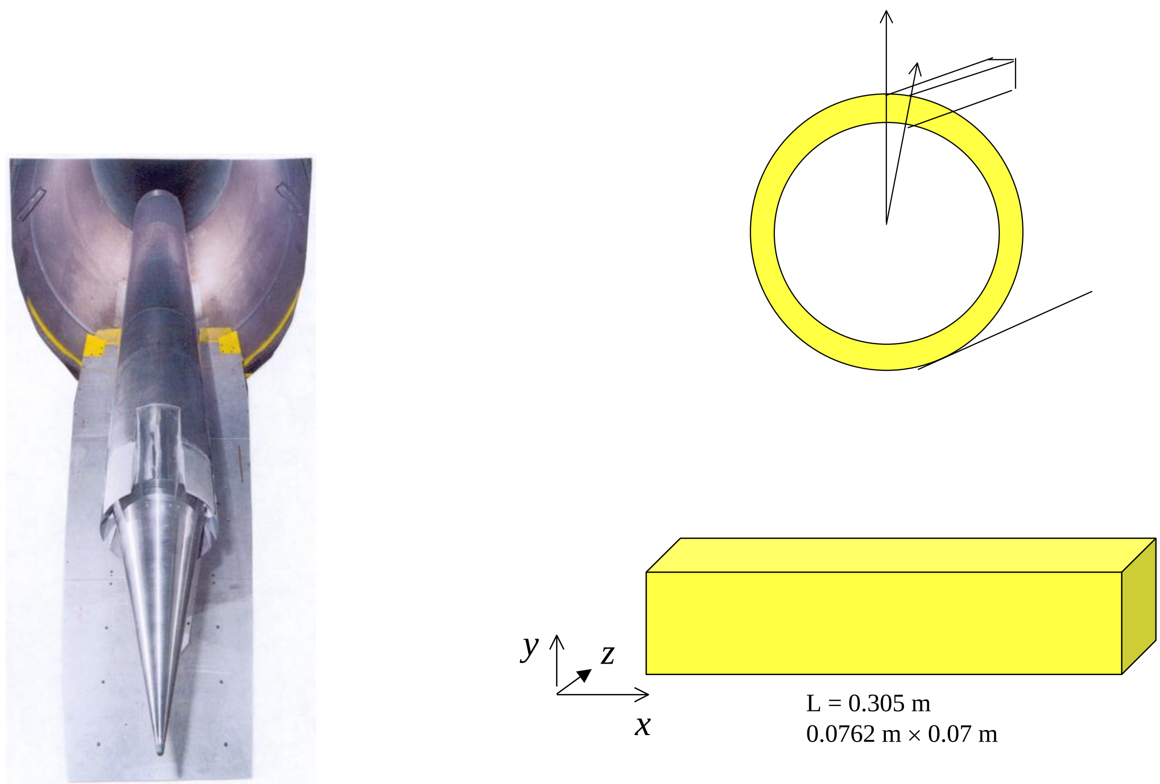


Figure 2. Scramjet Combustor Physical Model and Computational Domain.

The Viscous Upwind Algorithm for Complex Flow Analysis (VULCAN) code[17] was chosen for this study because it has demonstrated capability to model hypersonic turbulent reacting flows, incorporating multi-species, multi-reactions hydrogen-air finite rate kinetics models. The main mechanisms involved in fuel-air mixing are flow turbulence and molecular diffusion. VULCAN can model turbulent reacting flows over a wide range of conditions with a choice of several two-equation models such as Wilcox *k-w* model (1998 version) with or without the Pope correction term, and two-equation model of Menter based on Wilcox *k-w* model blended with Jones and Launder *k*-epsilon model.

The knowledge of the static temperature and of the species concentration in a reacting flow is indispensable for the understanding of the ignition and the reaction processes. VULCAN allows for an arbitrary chemical kinetic mechanism, specified as follows: The number of chemical reactions and the kinetic model data must be specified along with the number of chemical species. If finite rate chemistry is chosen, then the kinetic model data is specified through a kinetic model database file.

Several chemistry models are provided with the code. We used a 7 species/7 reaction hydrogen air kinetic model as the baseline. Although more comprehensive reaction models can be used, for this preliminary study in which we were interested in tracking mainly the production of the flame species H_2O and OH to determine the extent of combustion, this simple 7-reaction model is adequate. The Hydrogen-Air Reaction Mechanism used in this study is summarized below:

REACTION	REACTANT SIDE		PRODUCT SIDE
1	H2 + O2	<=>	2OH
2	H + O2	<=>	OH + O
3	OH + H2	<=>	H2O + H
4	O + H2	<=>	OH + H
5	2OH	<=>	H2O + O
6	H + OH + M	<=>	H2O + M
7	2H + M	<=>	H2 + M

We first analyzed many single-jet fuel injection cases, which served as calibration of the CFD code and provided the baseline data to scope the full simulations. The first 2-D analysis simulated the complexity of the interaction flowfield produced by a hydrogen jet injected into the cross flow of air at Mach 4.143 and 4.7, conditions representative of flight at Mach 10 and 12, respectively. Figure 3 shows the Mach number contours obtained for Case 1, compared with the schematic of the flow physics expected for transverse fuel injection flowfield. The results of those initial and other intermediate analyzes are reported elsewhere.[15]

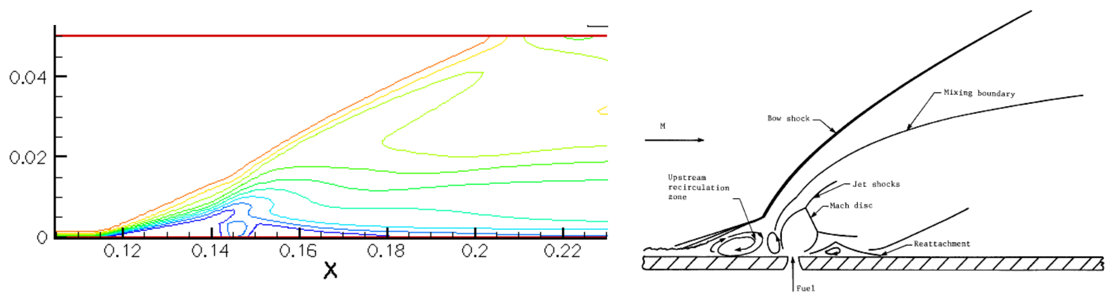


Figure 3. Mach Number Contours and Schematic of Flow Physics – Single Transverse Fuel Jet.[15]

We used Multiple Block Grid Generation Software (GRIDGEN) for the generation of 2D and 3D, multiple block, structured grids. We performed the post-processing of the data (visualization and quantification) with Tecplot 360.

The following sections summarize the most important results of the study. For clarity, the scope of the analysis is divided into 2 parts, namely, (1) 2-D Scramjet Combustor with Multiple Fuel Injectors; and (2) 3-D Scramjet Combustor with Multiple Injectors. Results obtained with cavity fuel injection is reported in Ref. [Musielak, 2006, and Musielak, HSABP TC Report 3, 2015], while Analysis of Vehicle Inlet Performance is reported in Ref [Musielak, HSABP TC Report 1, 2015] This distinction is made, as each study has its own objectives, computational grids, boundary conditions and computational approach, and therefore each study is considered to be independent from the others.

4. 2-D SCRAMJET WITH MULTIPLE FUEL INJECTORS

4.1 Introduction

A parametric analysis was first conducted by solving the two-dimensional/axisymmetric Navier-Stokes equations in order to find the effect of injector size, injector location, and multiple jet interaction on mixing and combustion performance. The first two cases analyzed consisted of an axisymmetric combustion chamber with one injector, as discussed in Ref. 17. The diameter of the fuel injector was varied to assess the effect of fuel/air ratio on combustion performance. As reported,[18] under these conditions ignition is likely to occur as soon as the H_2 fuel jet meets the high temperature oxidizer in the air stream. It was shown that the self ignition point is on the left side of the mixing layer around the jet (lean side), represented by both the large amount of H_2O produced and the rapid increase of temperature (indicative of heat release). These phenomena were observed in the initial simulations.[15] The second set of cases, reported below, was included to evaluate the effect of the fuel injection angle and to study the interaction of multiple fuel jets.

4.2 Physical Model and Boundary Conditions

To study the effect of multiple jet interaction on mixing and combustion performance, two injectors were positioned a few centimeters from each other. In an attempt to improve the level of fuel-air mixing, the injection angle was varied and the injectors were placed closer to each other. It was expected that the interaction of the two jets would enhance mixing and thus improve combustion performance. The injection angle was varied for this purpose as well. The computational domain (CD) is depicted in Figure 4. Its dimensions are: Length = 0.4572 m (18 in.), Radius = 0.05 m (1.96 in).

The half-width combustor was discretized with a single block computational grid having dimensions 201 × 153 (axial × radial). Grid cell size varies to provide proper resolution of flow gradients and to ensure the capture of the fuel/air interaction near the boundary layer. The grid spacing at the wall is 1.13429×10^{-5} m, which is correct for the use of wall functions.

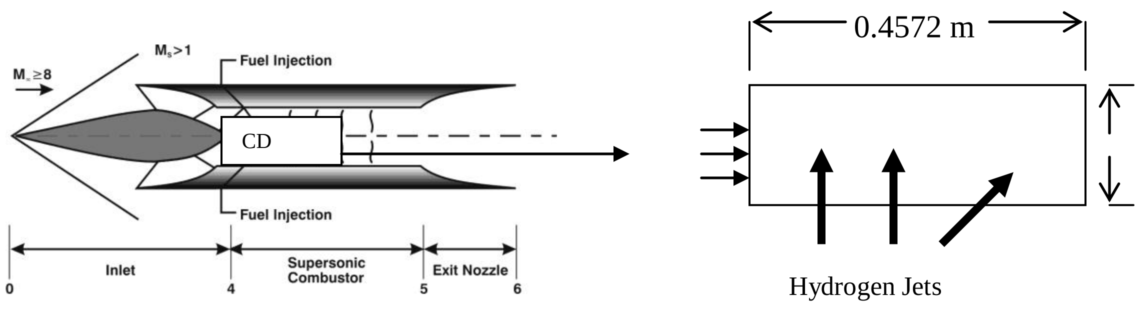


Figure 4. Computational Domain (CD) to simulate the 2-D Scramjet Combustor.

The airflow conditions at the combustor entrance (summarized in Table 1) were imposed on the inlet boundary. The airflow was assumed to contain only N_2 and O_2 with uniform mass fractions of 0.7686 and 0.2314, respectively. The conditions of the hydrogen H_2 fuel are summarized in Table 2. As shown for Cases 3, 4, and 5, two injectors are separated by a few centimeters and have different injection angles.

Table 1
Air Conditions at Scramjet Combustor Entrance

M_3	P_3 (kPa)	T_3 (K)	U_3 (m/s)	I_3
4.7	50.81	1300	3415	0.01

Table 2
Fuel Injection Conditions for 2-D Analysis

Case	P_j (kPa)	T_j (K)	d_j (mm) Inj1	x_j (m) Inj1	α_j Inj1	d_j (mm) Inj2	x_j (m) Inj2	α_j Inj2
3	309.75	290	2.9	0.023	90°	4.1	0.145	30°
4	309.75	290	2.9	0.023	30°	4.1	0.145	90°
5	309.75	290	2.9	0.023	90°	3.2	0.054	90°

Note: d_j = injector diameter
x_j = injector axial position
α = fuel injection angle with respect to x-axis

4.3 Results

Figure 5 shows plots of the percent fuel remaining in the chamber (at the exit plane) for different dual fuel jet configurations. As shown, mixing is greatly improved when a second injector is added: the percent of H_2 drops to as low as 27% for Case 5. This is not surprising, as the two fuel injectors are placed closer together (3.1 cm apart), and the interaction of the jets cause more vorticity in the flow, resulting in better fuel/air mixing.

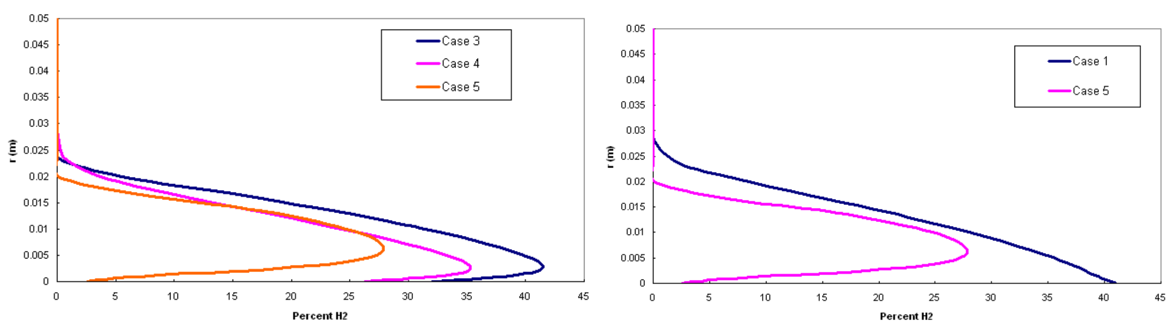


Figure 5. Effect of Fuel Jet Interaction on Mixing: (a) Two Fuel Injectors, (b) Comparison of Cases with One and Two Injectors.

This improvement in mixing due to jet interaction was also observed[17] when we compared Case 1 (baseline with one injector) and Case 5 (with two injectors close together). As a consequence of improved mixing, combustion efficiency is higher for Case 5.

Mixing efficiency η_m at a given cross section of the solution domain can be defined as the ratio between the mass flux of hydrogen fuel that would react to the mass flux of hydrogen fuel injected into the combustor,

$$\eta_m = \int_0^H \frac{m_{H_2}\rho u}{\rho u H_c} dy$$

where m_{H_2} is the mass fraction of hydrogen fuel that remains (unburned), and H_c is the height of the chamber.

Furthermore, the smallest amount of fuel that constitutes a meaningful presence of that fuel is derived from the definition of lean flammability limit. According to Kutschenreuter,[19] the theoretical lean flammability limit for hydrogen and air is 4% by volume, which translates into a mass fraction of 0.288%. From propulsive thrust considerations, a somewhat higher mass fraction may be appropriate. A value of 1% mass fraction (roughly ϕ = 0.34) of hydrogen in all forms is acceptable. Thus (for this study), the location where the mass fraction of H_2 reaches 0.01 is considered the point where all fuel has been consumed. Therefore, the indicator of fuel-air mixing is the mass fraction of fuel that is present across any plane of the solution domain, and the indirect assessment derived from the data in Fig. 5 is correct.

4.4 Discussion

The flowfield in the scramjet combustor is highly three-dimensional. The dominant flow structures observed in a *Jet in Crossflow* (*JICF*) configuration involving a sonic jet and supersonic crossflow are illustrated in the following schematic (Fig. 6), and the flow physics are explained below (not including ignition or combustion processes).

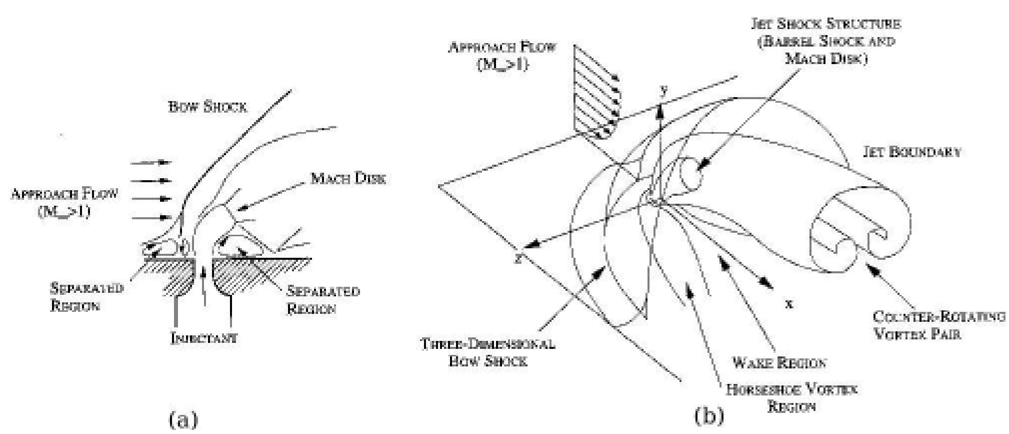


Figure 6. Schematic showing *Jet in Crossflow* (JICF) (a) side view; (b) perspective view.

In JICF, the adverse pressure gradient resulting from the jet interaction causes the turbulent boundary layer to separate and form recirculating zones upstream and downstream of the injection plane. The jet obstructs the supersonic crossflow and thus leads to the formation of a three-dimensional bow shock. The underexpanded sonic jet undergoes expansion, forming a Prandtl Mayer expansion fan and a system of incident-reflected oblique shocks. The barrel shock is the barrel-shaped structure formed due to expansion fans while the Mach disk is formed when the incident shock converges to a disk instead of a point. The horseshoe vortex and wake vortices are formed downstream of the injected jet. The counter rotating pair of vortices (CRPV) dominates the cross section of the jet in the far field and improves mixing by entraining the crossflow fluid.

Thus, we theorize that having more than one fuel jet in close proximity would result in increased vorticity, which in turn would improve the overall fuel/air mixing resulting in higher combustion efficiency of the combustor. In fact, it was just shown that, in the 2-D configuration, the addition of a second jet behind the main one clearly increases the mixing process. However, a simple 2-D simulation cannot account for the expansion and overall interaction of the jets that must occur in a full 3-D combustor configuration. Therefore, in order to determine how multiple jets in a combustion chamber affect mixing and combustion efficiency the next series of computational simulations must be done by solving the full 3-D governing equations, and considering a 3-D combustor geometry.

5. 3-D SCRAMJET COMBUSTOR WITH MULTIPLE FUEL INJECTORS

5.1 Introduction

Based on previous analysis, it is clear that the interaction of multiple fuel jets will improve the mixing process and thus can enhance the combustion characteristics of the Mach 4.7 combustor.

5.2 Physical Model and Boundary Conditions

To model the scramjet combustion chamber, a slice of the annular section is taken and assumed to be almost rectangular to represent the 3-D physical model for the CFD analysis. As shown in Fig. 7, the slice represents a segment of an annular combustor, which is treated as being an almost rectangular 3-D computational domain (the size of the annulus is much smaller than the diameter of the outer scramjet shell). The 3-D domain has the following dimensions: L = 0.3 m (12”), W = 0.0762 m (3”), and H = 0.070 m (2.75”). The computational grid has 129 × 97 × 65 grid cells configured to capture the most important features of the flowfield.

Hydrogen fuel is injected from three wall orifices, positioned in the chamber bottom floor at a distance of 10 cm from the entrance. As illustrated in Fig. 8, two fuel injector configurations were considered:

1) <u>One-Row Injection</u> - A row of three equally spaced fuel orifices traversing the bottom wall, at x = 0.1 m. Spacing between adjacent injectors is s/d = 1.63.

2) <u>Staggered Injection</u> - A row of two fuel orifices at x = 0.1 m, and a third orifice positioned 2.5 cm downstream. Spacing between adjacent injectors in the first row is s/d = 1.96, and spacing between the aft injector and the forward row is s/d = 3.7.

The computational domain was discretized with a single block grid having almost a million cells. Grid cell size was varied to provide proper resolution of flow gradients. The cells closest to the solid surface were set to be 2 x 10^{-6} m.

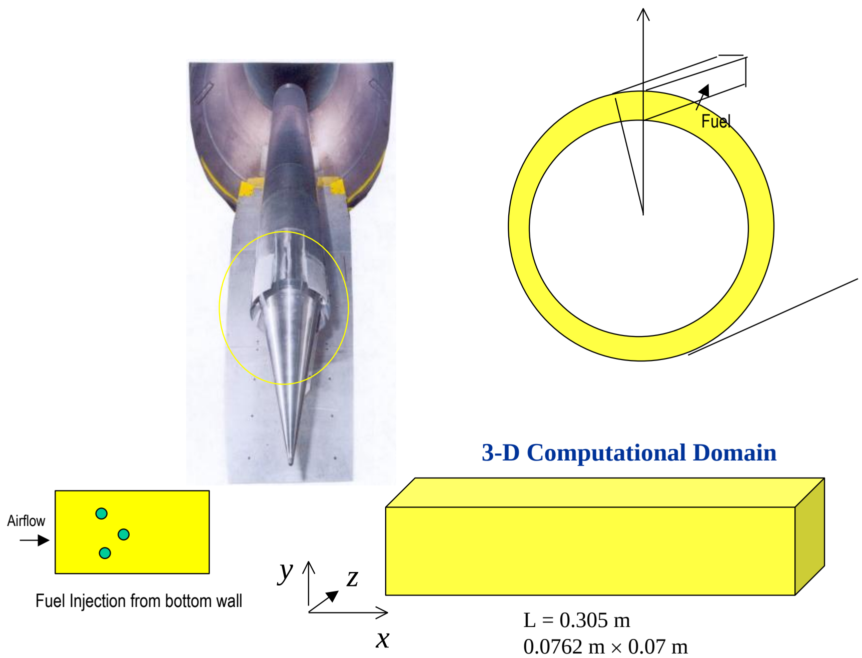


Figure 7. Annular Combustor Geometry: (a) Front View; (b) 3-D Slice Solution Domain.

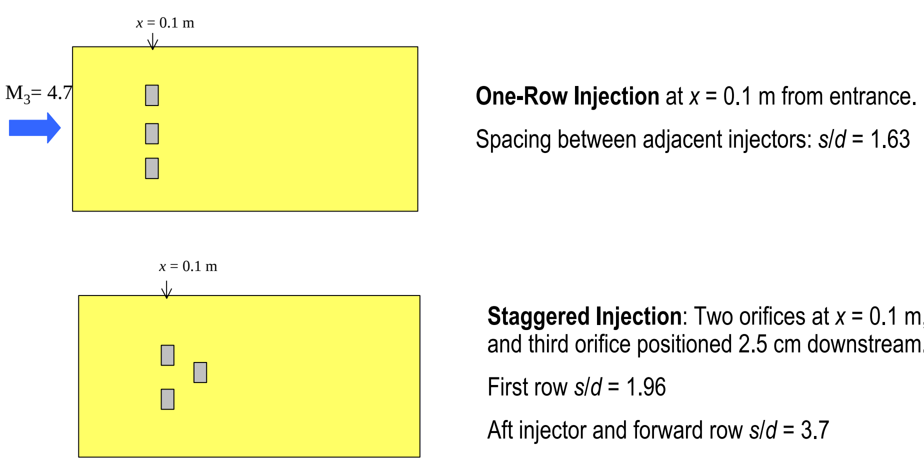


Figure 8. Multiple Fuel Injection Configurations.

All 3-D cases were simulated at the airflow conditions specified in Table 3 with the sonic hydrogen fuel injection conditions summarized in Table 3. The injection angle for the two cases is 90°. As summarized in Table 4, all cases were simulated with the airflow burner entrance conditions selected to represent a scramjet to power a vehicle at flight M_o = 12. No slip isothermal walls are assumed, using wall functions with a prescribed wall temperature. At the location of the injectors, fixed boundary conditions are imposed using the sonic hydrogen fuel static pressure and temperature conditions given in Table 4. The fuel injection angle is 90°. Based on unpublished preliminary analysis by Musielak[18] at the same burner conditions, using a lower injection angle resulted in lower mixing rates and much higher stagnation pressure losses, as compared with the cases reported herein.

The size of the fuel orifices was chosen to give mass flow rates yielding an overall equivalence ratio of 0.78. Supersonic outflow boundary with zeroth order extrapolation of all variables was specified at the exit of the chamber.

Table 3. Air Flow Conditions at Combustor Entrance

M_3	P_3 (kPa)	T_3 (K)	U_3 (m/s)	*Re*/m	*I*
4.7	50.81	1300	3415	9.2 x 10^6	0.01

Table 4. Hydrogen Fuel Injection Conditions

Case	M_j	P_j (kPa)	T_j (K)	ρ_j (kg/m^3)
1 and 2	1	310.33	350	0.30

High values of jet-to-cross-flow momentum flux ratio are typically used to achieve good penetration. The flow conditions selected for this study yield a value of J = 0.58.

5.3 Results

The near-field mixing of transverse jets is dominated by large-scale jet-shear layer vortices. In the cases investigated, due to the close proximity between injectors, the interaction of the fuel jets at each interface between pairs generates vorticity. This causes some regions of the flow to roll up into counter-rotating vortex pairs, which stir and mix the fuel with the high-speed air. The following sections describe these flow characteristics in more detail.

A. Flow Field Structure

The flowfield structure that develops from fuel injection through multiple orifices is very complex due to the interactions among the jets. Taking a slice of the flow along the midplane of the chamber, the Mach contours in Figs. 9 and 10 illustrate the general features of these flows (airstream direction is from left). The close-up of the axial velocity contours in the vicinity of the fuel injectors provides further details of the flowfield near the injection plane.

The *x*-*y* slice in Fig. 9 cuts directly through the middle of the third injector (center of the row, Case 1). Thus, the simulation shows the fluid mechanical features of fuel injected from a normal jet, e.g., the Mach disk, the interaction bow shock, the boundary layer separation and reattachment regions upstream and downstream from the injection point are clearly visible.

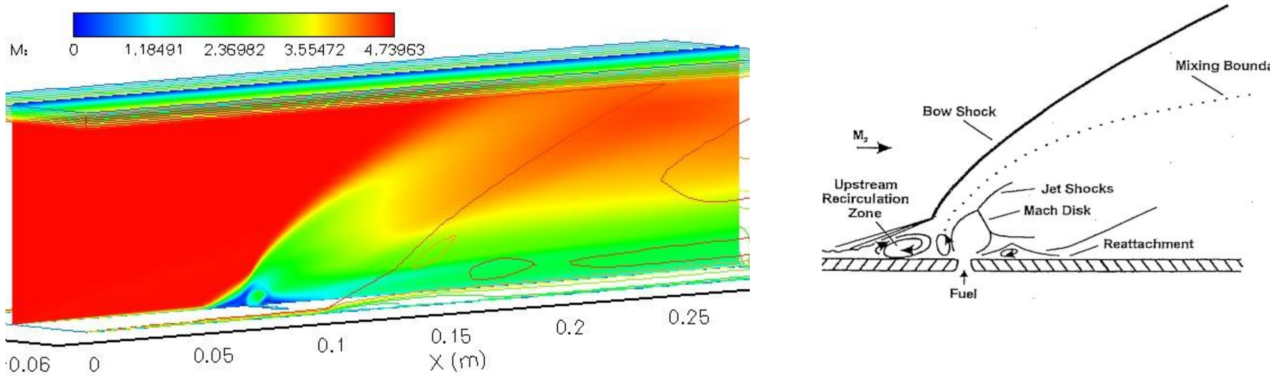


Figure 9. Mach Number Contours at Mid-plane, One Row Fuel Injectors.

On the other hand, with the staggered injectors (Case 2), a large region of separated flow between the forward and aft injectors appear to provide improved flameholding. Figure 10 illustrates the three dimensionality of the flow that results from the interaction of staggered jets. In this case, the *x-y* slice of the flowfield cuts through the spacing that separates the two leading jets and through the middle of the aft single jet. The main features are the bow shocks ahead of the jets, the Mach disk that forms over the aft injector, and the large zone downstream from the injection plane where recirculation stirs further the fuel/air mixture. In that region, the boundary layer is lifted considerably, causing the jet to move upward, hurling the mixing layer higher. The bow shock caused by the aft jet has a higher angle with the horizontal, causing it to interact with the shocks originated by the forward jets.

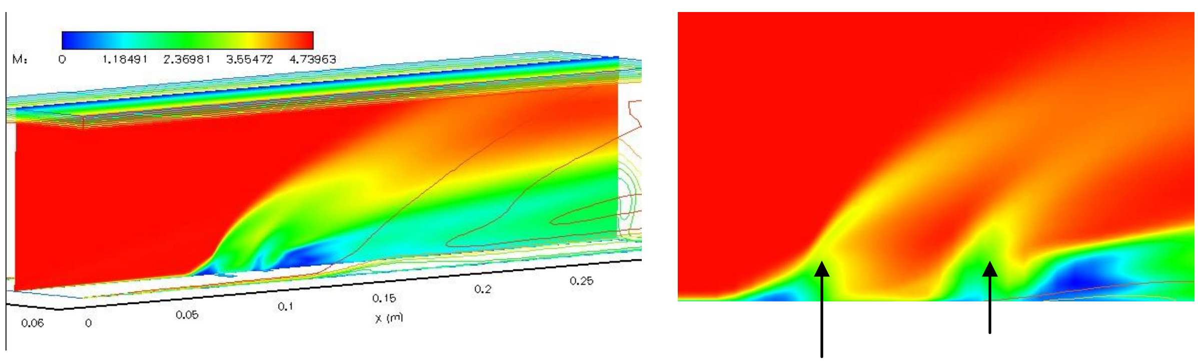


Figure 10. Mach Number and U-Velocity Contours at Mid-plane, Staggered Fuel Injection.

The most interesting feature of the 3-D flowfield with multiple injectors is the formation of vortex rings around the periphery of the fuel jets. As illustrated in the *y-z* slices of Fig. 11, in the region near the injector base, the injected gas moves with a higher velocity (tangent to the interface) than the free-stream air. As a result, large vortices are periodically formed engulfing large quantities of free-stream fluid and drawing it into the jet-shear layer (macromixing). These large-scale vortices have been observed with single jet experiments. In this study, a counter-rotating vortex pair (CVP) is observed, represented by the +/- vortices on each side of the row of injectors. As shown in Figs. 11 and 12, the contours of radial and tangential velocity indicate a relatively strong CVP.

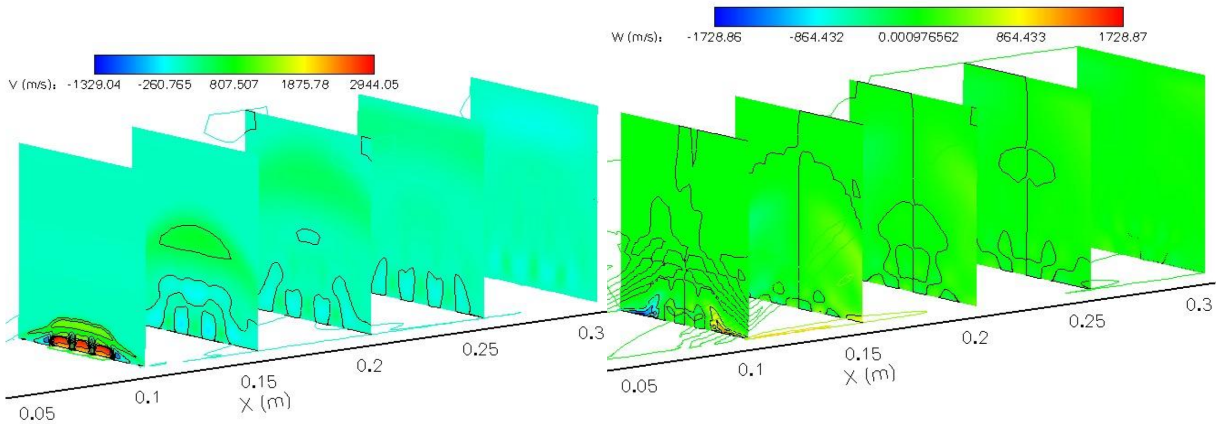


Figure 11. Radial and Tangential Velocity Contours – One Row Fuel Injection.

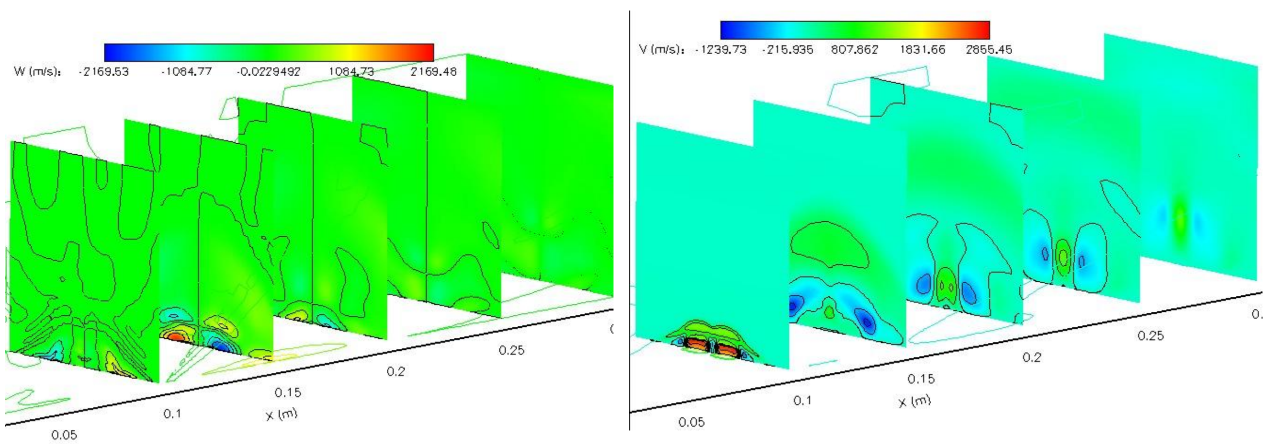


Figure 12. Radial and Tangential Velocity Contours – Staggered Jets.

B. Jet Penetration and Mixing

The interaction of the jets helps the fuel penetration of the three-orifice injection system. *Jet Penetration* is defined as the vertical height from the base of an injector to the edge of the mixing region where the fuel mass fraction is one half of one percent, given as y/d, where d is the diameter of the fuel injector.[20] (See Fig. 13 for an illustration of jet penetration).

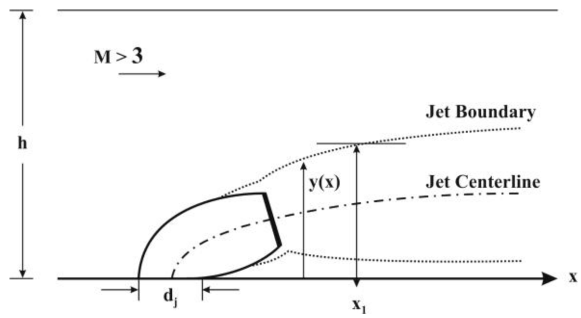


Figure 13. Geometry of Jet Penetration.

Contours of H_2 mass fraction distribution obtained with the two injection configurations are shown in Fig. 14. The *x*-*y* slices taken through the middle of the *z*-plane illustrate the degree of the jet penetration achieved. It shows the jet boundary for the plot on the right to reach a higher point, compared with the jet on the left (one-row aligned injectors). This suggests that the staggered injection approach has better penetration.

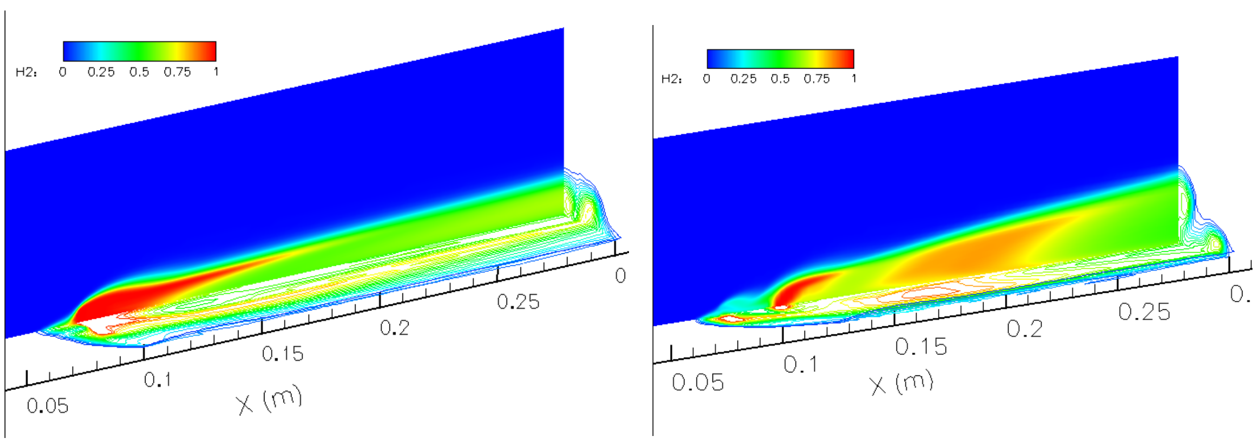


Figure 14. Contours of Hydrogen Mass Fraction, Aligned and Staggered Jets.

Figure 15 plots penetration profiles for the two injection cases at two axial stations along the combustion chamber. As shown, at the axial location that is 7.5 injector diameters downstream from the first row injection plane, $x/d = 7.5$, penetration is almost the same for both injector configurations. At this point, the fuel jets penetrate past the edge of the boundary layer up to a vertical distance from the bottom of the chamber that is about 10 times the boundary layer thickness, δ. However, as the jets expand downstream, the penetration characteristics change, and the staggered jets perform better. At the combustor exit plane, the edge of the mixing layer reaches $y/d = 3.4$ for Case 1, where $y = 12\ \delta$. With the staggered jets (Case 2) the mixing layer goes up to $y/d = 5.4$, where $y = 17\ \delta$.

The aft injector adds vorticity to the incoming fuel/air mixture that developed after the fuel was injected through the forward injectors. The shear action of the aft jet also tends to spread the gas out and away from the centerline producing additional mixing vortices. This effect is manifested in the reaction layer that develops in the lateral sides of the chamber.

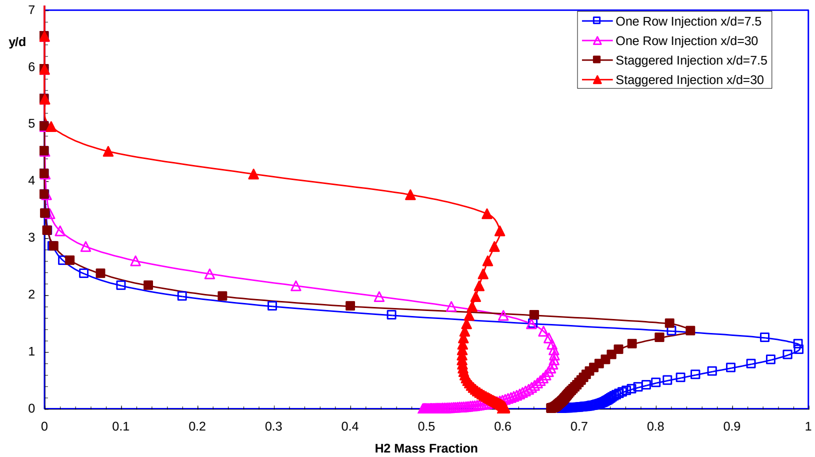


Figure 15. Penetration Characteristics of Multi Jets, Transverse Injection.

Mixing is also characterized by the rate of plume expansion in the chamber. For a lateral array of injectors, the axial distance where adjacent fuel plumes merge is an indicator of the expansion rate, as shown in the sketch below. From the plume expansion profiles shown in Fig. 16, it confirms that mixing is enhanced by the interaction of the closely spaced staggered jets, as the fuel plume expands less on the bottom wall of the chamber, at bottom of Fig. 16.

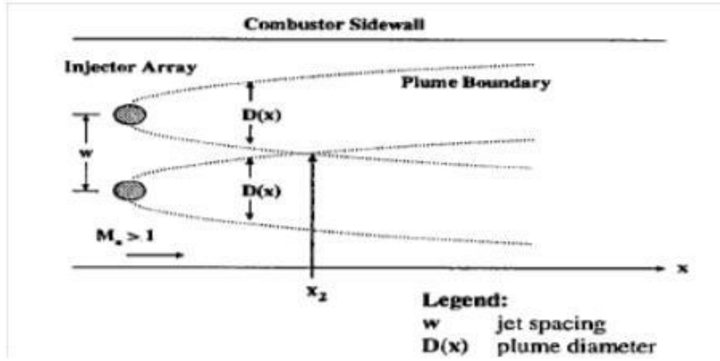
There are a number of empirical correlations for transverse jet penetration, obtained through the years by a number of investigators such as Rogers (1971), McDaniel & Graves (1988), Gruber, et al. (1995), and others.[21] Those correlations, although based on cold flow experiments, provide a measure of the penetration characteristics of transverse jets that is a function of the momentum jet to airstream ratio, as summarized in Fig. 17 from by Gruber, et al.[21] Selected empirical correlations used in this study are given below.

Rogers (1971) $$\frac{y}{d} = 3.87\, J^{0.3} \left(\frac{x}{d}\right)^{0.143}$$

McDaniel & Graves (1988) $$\frac{y}{d} = J^{0.344} \ln\left[2.08\left(\frac{x}{d} + 2.06\right)\right]$$

Gruber, et al. (1995) $$\frac{y}{d_{eff} \cdot J} = 1.23\left(\frac{x}{d_{eff} \cdot J}\right)^{0.344}$$

We compared the penetration data obtained in this study with above correlations. We found that the correlations of Rogers, and McDaniel & Graves provide upper and lower bounds for our results, as shown in Fig. 18. A closer match is not possible, as the CFD data incorporate chemical kinetics, effects which are not accounted for in the above empirical correlations.



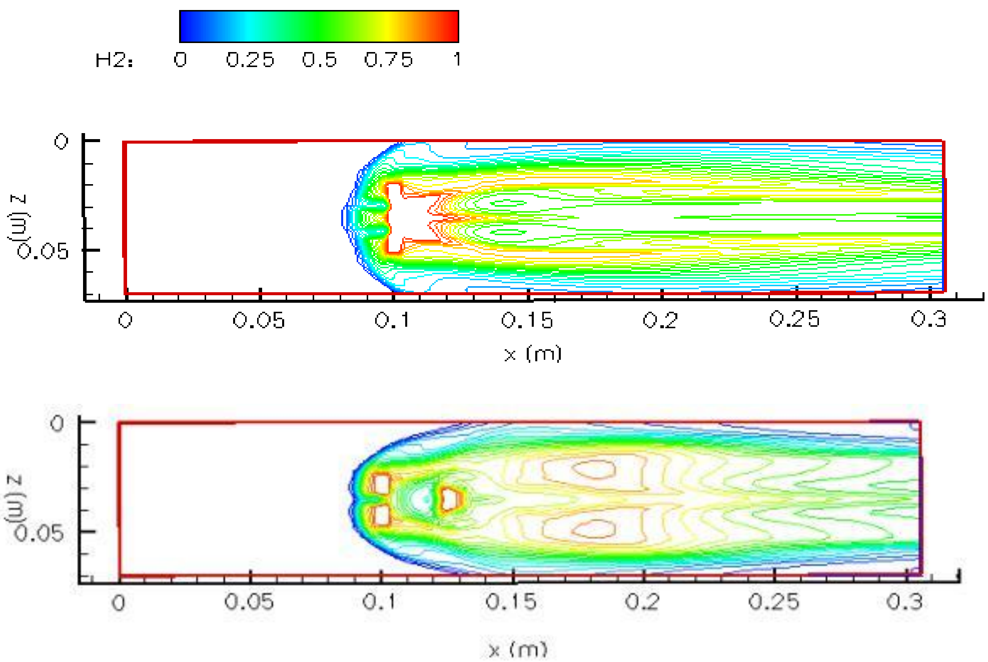


Figure 16. Plume expansion characteristics (a) One-Row Injectors, (b) Staggered Injectors.

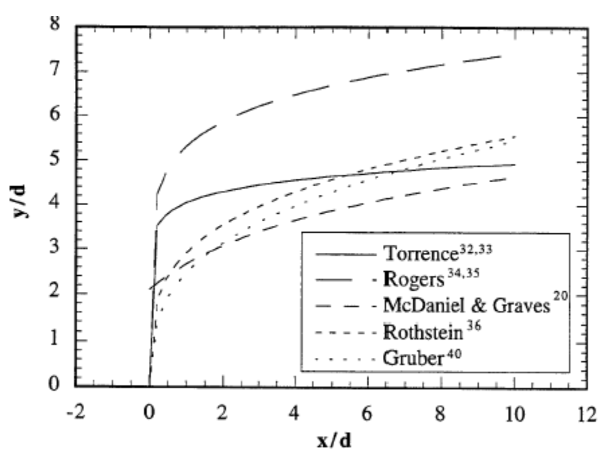


Figure 17. Transverse Jet Penetration Correlations (From Gruber, et al. WL-TR-96-2102).[22]

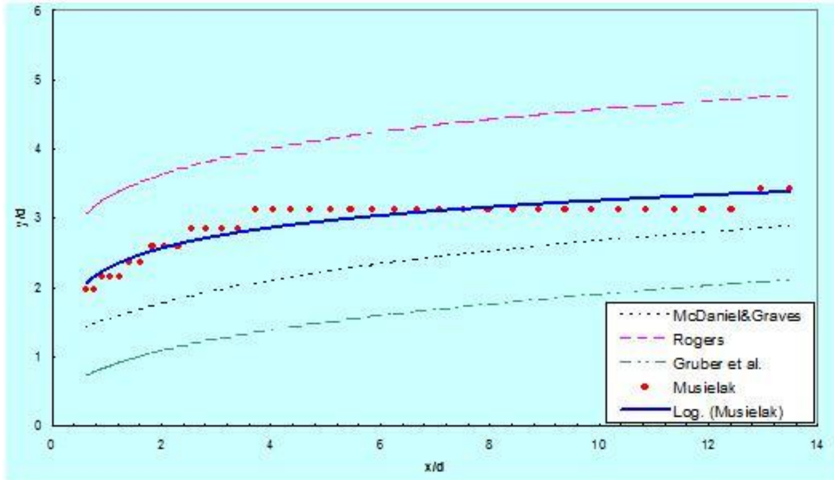


Figure 18. Comparison of Jet Penetration CFD Results with Empirical Correlations.

Fuel Distribution and Overall Combustion Characteristics

We observe that the fuel jets penetrate the boundary layer and react with the high velocity airstream in a mixing layer above the edge of the fuel jets. To compare the extent of mixing from the two injection configurations, the contours of H_2 and H_2O mass fraction shown in the *x-z* slices of Figs. 19 and 20 are examined as flow moves along the chamber. The shape of the fuel distribution at each plane is indicative of the mixture uniformity and combustion characteristics. The distribution of H_2O indicate the degree of reaction taking place as the fuel jet is mixed and reacts with the oxygen in the airstream. As shown, the reaction zone follows the shape of the fuel/air mixture distribution. Note the height of the fuel signature for the staggered injection case, indicative of its higher penetration and stronger reaction but less uniformity across the chamber, as compared with the one-row injection case.

The extent of reaction is also given by the increase of temperature that is 2.5 times the burner inlet temperature ($T = 2.5\ T_3$) with staggered injection, and 2.4 T_3 with the row of aligned jets (See Fig. 21). A degree of lateral flow spreading and mixing is also observed in the two cases.

To approximate the variation of temperature with ϕ, a simplified 1-D enthalpy balance between the fuel and the air can be performed. The gases are assumed to be thermally and calorically perfect and ideally mixed. Based on this model, the enthalpy balance between the air and fuel streams can be written as $m_a c_{p_a}(T_a - T_m) = m_{H_2} c_{p_{H2}}(T_m - T_{H_2})$, where m_a and m_{H_2} are the mass of air and H_2, respectively.

The mixture temperature T_m is defined as

$$T_m = \frac{T_a + \alpha\, T_{H_2}}{1 + \alpha}$$

where

$$\alpha = \frac{m_{H_2} c_{p_{H2}}}{m_a c_{p_a}} = f\,\frac{c_{p_{H2}}}{c_{p_a}} \qquad (f = \text{fuel} - \text{air ratio})$$

For the case under consideration, with $T_a = 1300$ K and $T_{H_2} = 350$ K, the mixture temperature is $T_m = 1052.5$ K at stoichiometric conditions, $f_{st} = 0.0291$.

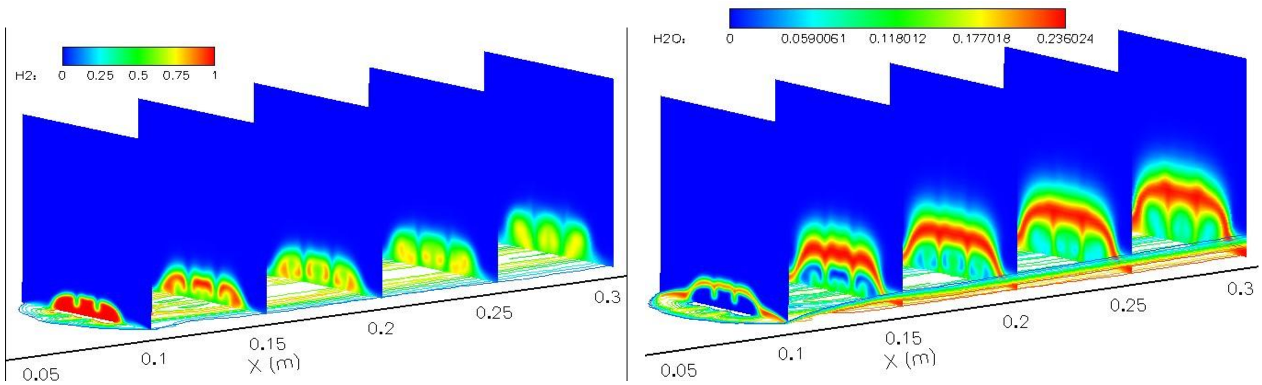


Figure 19. H_2 Fuel and H_2O Mass Fraction - One Row Aligned Orifices.

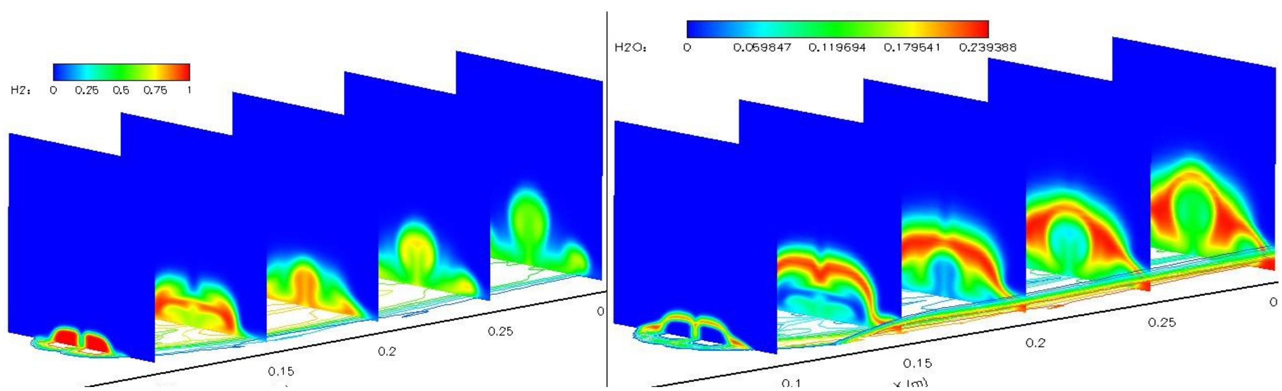


Figure 20. H_2 Fuel and H_2O Mass Fraction - Staggered Orifices.

The variation in ignition time with fuel/air equivalence ratio for cold H_2 ($T_{H_2} = 300$ K) injected into hot air is given in Fig. 21. As shown, the ignition times for the cases under consideration are on the order of 300 μsec at an equivalence ratio of 1, but it drops considerably to less than 70 μsec for the case of a lean fuel-air mixture. Thus, a trade-off between mixing, ignition, and equivalence ratio must be made in order to maximize the parameters that result in the highest combustion efficiencies.

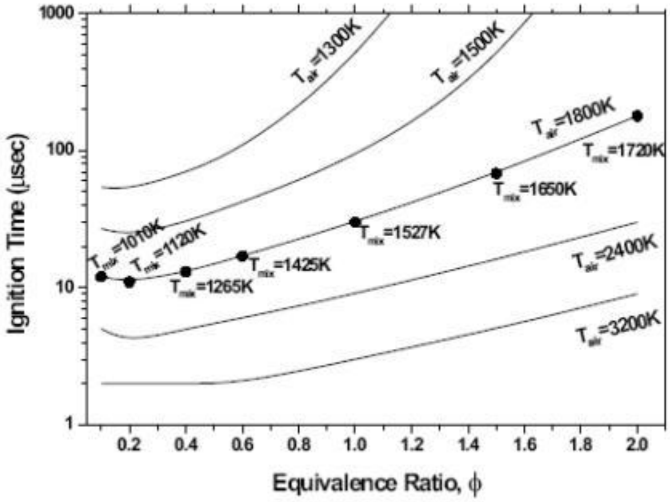


Figure 21. Variation of ignition time with equivalence ratio for cold (300 K) H_2 injected into hot air.

Stagnation Pressure

Overall, the stagnation pressure losses were determined to be lower for the staggered jet injection configuration. The wall pressure profiles computed along the midplane of the combustor slice are depicted in Fig. 22. The difference in the pressure profiles for the two injection cases results from the difference in the flow structure observed in Figs. 10 and 11, where the staggered case showed a large recirculation between the forward jets and the aft injector. As shown, for this case the dip in the pressure (red curve) occurs at $x = 0.132$ m, which is 1.3 jet diameters downstream from the aft jet. However, the boundary layer reattaches effectively 3 centimeters downstream from the last injector. At the combustor exit plane, the one row injection configuration has a pressure ratio $P_w/P_3 = 0.81$, while the staggered case has a value of $P_w/P_3 = 0.90$.

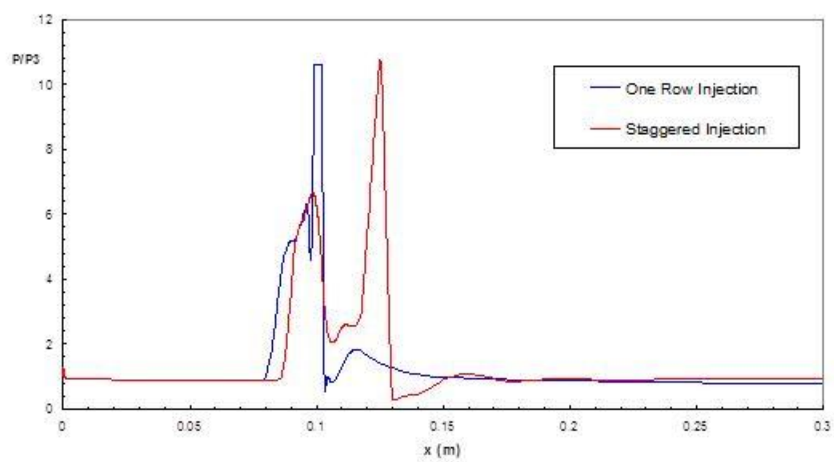


Figure 22. Wall Pressure Distribution Along the Combustor Centerline.

There is a significant variation of temperature and equivalence ratio ϕ inside a supersonic combustor since a relatively cold fuel is injected into hot, high-speed air. As noted above, the variation of temperature through the mixing layer around the fuel jet is represented by the mixture temperature T_m. The self-ignition temperature of hydrogen is 823 K; the mixture temperature must be greater than this value to ensure autoignition of the H_2 jet as it meets the air stream. Since the temperature of the mixture will be higher at low values of ϕ, and since ignition time is a strong function of T_m, it is expected that the self-ignition point will be on the lean side of the mixing layer. This is represented in Fig. 23, where the rapid increase of temperature (representative of heat release) is found on the left side, over the barrel shock that forms around the fuel jet.

Research published by Lee[23] confirmed the findings reported herein and are illustrated in Figs. 14 and 23 (b). In Lee's paper, the mixing characteristics of two jets (one behind the other) are studied computationally. The effects of jet-to-cross-flow momentum flux ratio (*J*) and the distance between the two injectors on mixing are included. It was shown that the mixing characteristics of the dual injection system are very different from those of a single jet (our theory). Lee found that the rear jet is strongly influenced by blockage effects due to the momentum flux of the front injection flow and thus has higher expansion and penetration than the front jet. He also found that the dual injection system has a higher mixing rate and a higher penetration but has more losses of stagnation pressure than the single injection system.

Figure 23(c) shows the comparison of our results with the schematic illustrating the flow physics of the dual transverse injection system. Note the higher fuel penetration of the rear jet. In both 2-D and 3-D cases, the jet interaction resulted in a stronger mixing layer.

6. CONCLUSIONS

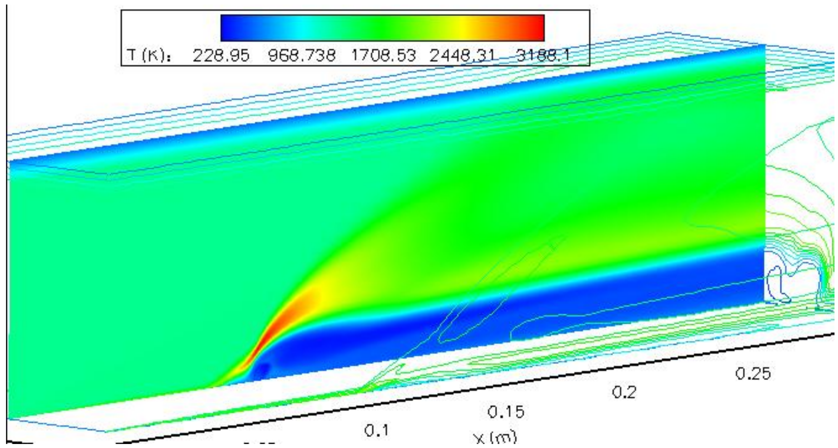
Jet penetration, mixing characteristics, and degree of reaction show a strong correlation to the degree of interaction among the jets. Staggered jets produced better penetration, as compared to the case when jets were aligned in a straight row. The following summarize the main conclusions of this study.

- **2-D Combustor with Multiple Fuel Injectors**

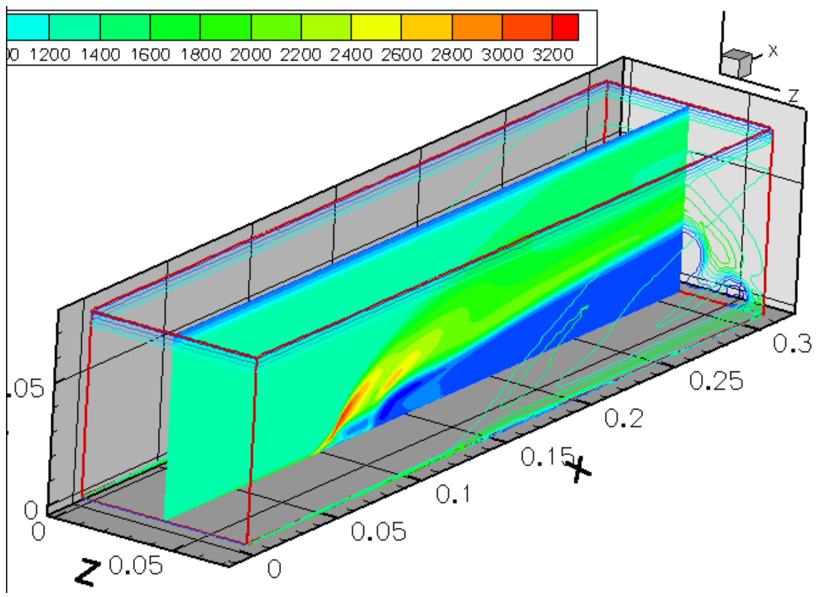
For the 2-D combustor analysis the effect of multiple fuel injection in a fixed-diameter computational domain was studied with two H_2 jets. The effect of injection angle, injector diameter, and position with respect each other was studied. It was found that increased burning resulted when two injectors were positioned close to each other.

- **3-D Combustor with Multiple Fuel Injectors**

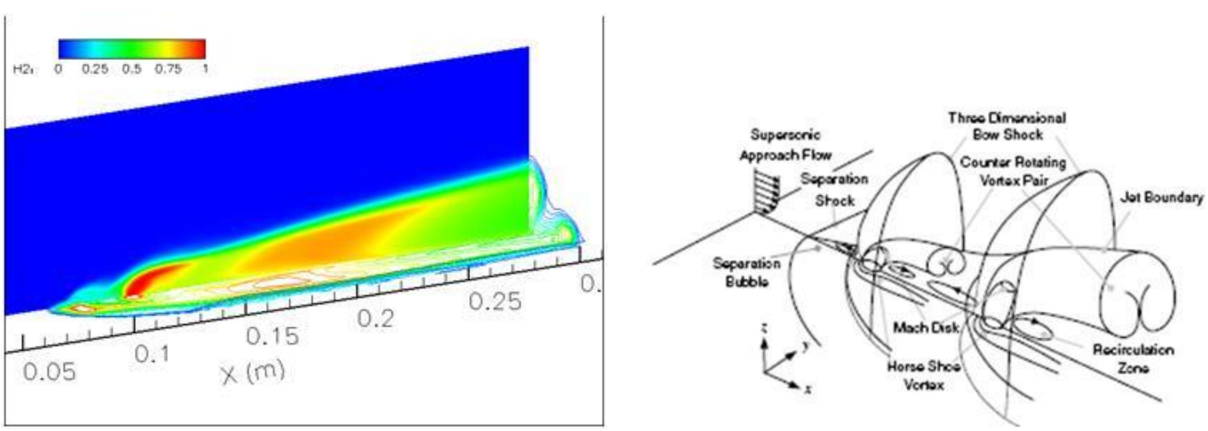
For the 3-D combustor analysis the effect of fuel jet interaction was studied by comparing a configuration with three fuel injectors on a row, as well as a three-staggered injection scheme. The 3-D flowfield that results from H_2 fuel injection through multiple wall-mounted orifices was characterized effectively. The complexities of the high-speed reacting flow revealed significant interaction by the proximity of the fuel jets that promoted mixing with the Mach 4.7 airstream.



(a)



(b)



(c)

Figure 23. Temperature Distribution along Combustor *x-y* Plane (a) One Row Injection; (b) Staggered Injection. (c). Flow physics of the dual transverse injection system.

Jet penetration, mixing characteristics, and degree of reaction were found to be directly related to the degree of interaction among the multiple jets. The case with staggered injection had better penetration as compared with the one row injection.

At the conditions of a Mach 12 flight vehicle, the extent of fuel-air mixing has yet to be maximized. Thus, although the reactive mixture in the chamber autoignites and the interaction of multiple fuel jets promotes penetration and mixing, the injection conditions must be optimized to increase the level of mixing achieved with this approach. Therefore, the next phase of this study included evaluation of multiple injectors in a diverging combustion chamber with the same air flow conditions at the entrance, but with a combination of wall cavities and staggered injector configurations. An assessment will be made to the extent of fuel-air mixing that can be achieved with these variations and their effect on combustion efficiency. The original plan was to develop design criteria for the Mach 10-12 scramjet engine, which would include the arrangement of fuel injectors that yields the shortest mixing length and highest combustion efficiency.[24] At the same time, the requirement for uniformity of fuel distribution and minimum pressure losses was to be imposed in the selection of best fuel injection configuration. Some promisiong results were obtained with the added flame holding capability that results from the use of wall cavities.[25]

7. RECOMMENDATIONS FOR FUTURE WORK

The results obtained in this study are deemed to be a valuable contribution to the Hypersonics R&D Program. To ensure that the data support the development of a scramjet engine, additional studies and assessments of critical enabling technologies must be included in the program. This requires more research and modern codes capable of modeling the complex flow processes found in scramjet systems.

A study focused on the design of a Mach 10-12 flow path is necessary but *not sufficient* for the development of a scramjet engine as an integral part of a hypersonic missile system. Although the limited analysis of the propulsion subsystem to date is encouraging, a comprehensive design effort and more analysis over a representative range of operating conditions are necessary to determine the engine's operability within the constraints imposed by a given concept.

The industry and government together have established road maps to reach initial operational capability. A road map should include six phases: (1) system specification development; (2) system concept development; (3) technology risk reduction; (4) prototype design and test plan; (5) engineering and manufacturing development.

Finally, to support any scramjet design and hypersonic wind tunnel design and development efforts we recommended to acquire a suite of CFD codes plus optimization and finite element modeling (FEM) tools adequate to perform the increasing complex calculations and data analysis. Having a wide array of tools will ensure that more technical aspects of scramjet design and development will be analyzed and better understood.

To continue building a comprehensive, long-term, far-reaching Scramjet Technology Development Program it is recommended that future work include the following studies:

- **Comparison of CFD Simulations with Experimental Data** – A benchmark experiment is needed to validate the CFD simulation of this study. In order to increase the confidence of these results, it is recommended to compare them with available test models. The entire flowfield in a scramjet combustor is very complicated and involves transient reacting flows with numerous fuel jets injected into the air crossflow. The fuel/air mixing itself involves separation, shocks and vortex systems. This process is further complicated by ignition and flame transport across the combustor.

- **Hydrocarbon Fuel Injection System** – Study scramjet fueled with hydrocarbon fuels, and develop a hydrocarbon fuel injection system suitable for hypersonic missiles in the Mach 8 to 10 range. This study will add depth to the Scramjet Development Program and it will increase our level of competence to pursue funding from other DoD organizations currently developing hydrocarbon-fueled scramjets.

- **Modeling and CFD Simulation of Hypersonic Wind Tunnel Test Section –** In order to predict the performance capabilities of the Hypersonic Wind Tunnel, a study must be carried out to model the flowfields developed in the test section. Issues of flow quality and operational limits can be addressed before the tunnel is completed.

- **Study of Scramjet Scaling –** Carry out a study to determine the scaling laws for the performance of scramjet propulsion. This is particularly important if the wind tunnel will be limited to test subscale models.

- **Study of Advanced Materials for Hypersonic Applications –** The ability of the cooled flow-path components to operate over 1000°F hotter than the state-of-the-art metallic concepts will add system design flexibility to scramjet vehicle concepts.

We elaborate further on these recommended topics, for they serve as a starting point for new research projects.

1. Comparison of CFD Simulations with Experimental Data

Along with theory and experiments, computational fluid dynamics (CFD) is a fundamental and powerful tool used in hypersonics research to enhance understanding of complex flow processes. Experimental data is the observation of the "real world" in some controlled manner. By comparing the CFD results to experimental data, one hopes that there is a good agreement, which increases confidence that the physical models and the code represent the "real world" for this class of simulations. However, it should be noted that the experimental data contains some level of error. This is usually related to the complexity of scramjet experiments. As explained earlier, the flowfield is very complicated

When simulating the scramjet reacting flowfield, the computational models to be validated will include boundary conditions, diffusion and viscous models, a temporal algorithm, turbulence models, and capability to resolve vortex dynamics, shock interaction, and compressible mixing. To test these models, two different physical problems can be reproduced: Richtmyer-Meshkov Instability and Jet in Crossflow.

Ritchmyer-Meshkov Instability is a shock-accelerated instability that involves mixing of two gases. The mixing mechanism is dominated by shock interaction and vortex dynamics. This is a standard problem for CFD validation. Two-dimensional transient simulations of this instability can be used to compare with the experiments.

A sonic jet into a crossflow is a special case of the Jet in Crossflow (JICF) problem. Reliable experimental data for JICF cases is available for comparison with numerical simulations of a flow similar (but not identical) to those in this study. The mixing in JICF is dominated by shock and vortex systems. JICF is highly three-dimensional and may be turbulent.

Together the simulations of Richtmyer-Meshkov Instability and Jet in Crossflow can help validate the computational models of VULCAN responsible for predicting the mixing of fuel jets injected into the supersonic crossflow of the scramjet combustor. The ignition and chemical kinetics will have to rely on other relevant benchmark data found in the literature.

3. Hydrocarbon Fuel Injection System

Fuel is becoming the integrating factor of the complete hypersonic vehicle system. It is difficult to give a reasonable estimate for a long-term endothermic fuels plan without bounding the problem, in that key drivers and constraints for fuel and fuel system development come from other nominally distinct disciplines—thermal management, combustion, and ground handling, to name a few. Thus, the fuel ultimately selected for a given hypersonic vehicle will probably be a compromise between cost, operability, and performance in a number of areas. Concepts for large vehicle for air-breathing space access are fueled by liquid hydrogen, which has unmatched combustion performance and heat sink capability. Yet, liquid hydrogen's low density ($1/11^{th}$ that of hydrocarbons) and high handling cost for maintaining it at approximately 20 K leave plenty of room for alternative fuels for many applications that are incompatible with large (hydrogen) vehicles and a "hard cryogen" infrastructure.

A long-term fuels research program should be part of a scramjet-powered vehicle development, since the vehicle mission and operational envelope drive fuel selection. Moreover, since some hypersonic vehicles will be powered by rocket-based propulsion systems, the study of fuels would also benefit from coordination with the development of rocket propellants. The three main vehicle categories—expendables (missiles), accelerators (launch vehicles), and cruise—place quite different constraints on the fuel. In general, research is urgently needed for techniques to remedy the inherent shortcoming in hydrogen as a fuel for high Mach tactical applications.

The physics of the fuel behaviour are of great importance for hypersonic vehicle applications. The heat sink capacity of a fuel—hydrogen, for example—is due to its sensible heat and is thus proportional to the maximum temperature the fuel can achieve and the magnitude of its specific heat. Hydrogen has a much higher specific heat than hydrocarbon fuels, and assuming that the temperature change ΔT is the same, is considered to have a higher heat capacity than hydrocarbon fuels; however, less hydrogen mass is required to achieve a given amount of heat release, so the hydrogen fuel flow rate is lower than that of a hydrocarbon. A more relevant measure of heat sink potential is the ratio of specific heat to the heat of combustion, which brings hydrocarbon fuels more in line with hydrogen in terms of heat capacity (within a factor of 2 or 3). Thus, it is not so much the specific heat of hydrogen that gives it better heat capacity than hydrocarbon fuels but rather its ability to sustain higher temperatures and thus increase the ΔT part of its heat capacity. In the absence of other considerations, conventional hydrocarbon fuels cannot be driven to as high a temperature as hydrogen, which is capable of sustaining temperatures up to the operating temperature of the engine.

In the case of an endothermic fuel like JP-7, another source of heat capacity is made available by cracking the fuel, a process of breaking its long-chain hydrocarbon molecules into lighter molecules that absorb heat (an endothermic process). In an endothermic fuel, the heat sink capability of the fuel is made up of its sensible heat plus any net endothermic capacity derived from high fuel dissociation reactions. The key is to find a fuel that has endothermic capability without degrading its exothermic capability. Typically, hydrocarbon decomposition processes are accompanied by carbon formation, or coking. Coking tends to foul heat transfer surfaces, which is undesirable. Thus, there are two parts to calculating the upper limit of a hydrocarbon fuel's heat sink capability: the maximum temperature achievable without the system coking up, and the endothermic capacity of the cracking reactions that can occur.

Current U.S. state-of-the-art heat sink capability for an endothermic liquid hydrocarbon fuel under realistic conditions is roughly 1,500 Btu/lb, with a limit of 1300°F. Under operational conditions, a 1300°F temperature limit allows for a coking-limited lifetime of approximately 15 min. In this case, the total fuel heat sink capacity is a combination of 500 Btu/lb from endothermic reactions and roughly 1,000 Btu/lb from sensible heat. If the coking limit of a fuel could be raised by a few hundred degrees, its heat sink capability could be enhanced.

It is important to point out that the endothermic heat capacity of 500 Btu/lb is much less than the theoretical heat capacity achievable if the most desirable fuel reaction products could be obtained. Cracking JP-7 to 100 percent ethylene would absorb 1,500 Btu/lb, versus the 500 Btu/lb obtained from the product mix of methane, ethylene, ethane, etc. Thus, there is clearly room for increasing the endothermic capability of even our present candidate fuels. Because of the endothermic capability of our present hydrocarbon fuels, at the upper Mach number range for hypersonic air-breathing flight (~ Mach > 8), hydrogen is the fuel of choice; however, because of hydrogen's difficulty of storage and its limited volumetric energy density, hydrocarbon fuels are more practical for lower Mach number applications.

All of the hydrocarbon-based vehicles contemplated to date anticipate using JP-7. A robust fuels research and development (R&D) program could lead to higher Mach number limits for hydrocarbon fuels. The Russian AJAX (concept) vehicle, for example, increases heat sink capability by an interesting variation on conventional endothermic fuels. In the AJAX concept, water is added to the fuel to achieve steam reforming. In essence, steam reforming is fuel + water → $CO + H_2$. This reaction absorbs (theoretically) 2,400 Btu/lb (versus 1,500 Btu/lb for cracking to ethylene). This is how the Russians theoretically obtain Mach 10 capability for AJAX, although the concept suffers reduced range owing to water consumption in the propulsion cycle.

In the 1960s and early 1970s, the Air Force funded a considerable effort at Shell Research to develop endothermic methylcyclohexane (MCH), based on studies that identified the heat sink required for hydrocarbon-fueled hypersonic vehicles. Endothermic MCH required a supported platinum catalyst for dehydrogenation (to toluene and hydrogen) that was developed in pellet form (as is used industrially in fluidized beds for petroleum processing). After a long period of dormancy in the 1970s and 1980s, the work was restarted by Allied Signal (now Honeywell) and culminated in an expendable turbine-engine test, where the fuel was used to cool a hot air stream and then burned in an engine. There were two drawbacks to this technology: (1) regenerative cooling is best accomplished through a wall-mounted catalyst rather than pellets and (2) MCH is a relatively expensive specialty fuel.

In the mid-1980s, the Air Force Propulsion Directorate led an effort to develop the endothermic potential of thermally/catalytically cracked liquid hydrocarbons using commercially available, wall-mounted zeolite catalysts. This effort first reported the endothermic potential of JP-7 and JP-10, as well as JP-8. The resulting endothermic hydrocarbon fuel capability contributed to selecting such fuels for the HyTech engine development program, which began in the mid-1990s. This fuel development effort has continued sporadically since HyTech funding became available. Recent tasks include extending the endothermic heat sink database to higher flow rate conditions and to other fuels (e.g., RP-1), as well as looking at relative combustion performance of the various alternative fuels. The fuels research effort has also looked at the applicability of endothermic fuels to reusable aircraft and other applications.

To extend the Mach number capability of non-hydrogen-fueled hypersonic vehicles, one has to either improve the fuel heat sink capability or reduce the heat load from the vehicle into the fuel. The use of high-temperature ceramic engine structures appears to be capable of significantly reducing the required engine cooling (since the heat load into the fuel is proportional to the $T_{combustion}$ less the $T_{surface}$ driving force). Two key issues need to be addressed: (1) the structural manufacturability and durability of ceramic structures and (2) exposure of fuel to high-temperature surfaces and the impact on coking. (This can be combined with research on advanced materials, as recommended below.)

Another research study should look at more exotic alternatives, such as liquid metal decomposition and high-energy-density fuels. The fuel development should include both combustion and regenerative cooling research.

4. Modeling and CFD Simulation of Hypersonic Wind Tunnel Test Section

Adequately modeling the flow characteristics inside the test section (after expansion through the facility nozzle) will be a challenging task, as the multi-phase flow will be influenced by strong shocks, chemical kinetics, and viscous interactions. The length scales associated with some of these flow phenomena could be several orders of magnitude smaller than the size of the proposed facility. Therefore, the CFD code selected to carry out this task must solve the Navier-Stokes equations and include enough sophistication to simulate the complex physics of the flow, including multi-phase, diffusion, turbulence, and heat transfer models. Furthermore, to simulate the flowfield in the test section, the computational grid resolution must be sufficiently fine to resolve the small-scale physics.

In order to simulate hypersonic flight on the ground we must duplicate the freestream properties. By matching various similitude parameters, accurate simulation of certain phenomena can be conducted in test conditions that differ from the true flight conditions. Such parameters include the Reynolds Number, the hypersonic similarity parameter ($M\Theta$), the binary scaling parameter (ρl), the product ($\rho^2 l$), and the matching of the total enthalpy (H_0). Furthermore, since the size of the test section will invariably be small, it will only allow testing of subscale models. When subscale models are tested, certain scaling laws are used in an attempt to compensate for the smaller model. A study of scramjet scaling is therefore also recommended, as described below.

5. Study of Scramjet Scaling

Techniques for engine scaling over two orders of magnitude are being developed throughout the hypersonics community. Thus, it is recommended to carry out a scaling study to assess subscale models effects and establish scaling laws to improve the scramjet engine operating margin.

6. Advanced Materials for Hypersonic Applications

Lightweight, high-temperature, actively cooled structures have been identified as a key technology for enabling reliable air breathing hypersonic propulsion. Actively cooled carbon and CMC structures may meet high-performance goals at significantly lower weight, while improving safety by operating with a higher margin between the design temperature and material upper-use temperature. Studies have shown that using actively cooled CMCs can reduce the weight of the cooled flow-path component from 4.5 to 1.6 lb/ft^2 and the weight of the propulsion system's cooled surface area by more than 50 percent. This weight savings enables advanced concepts, increased payload, and increased range. The ability of the cooled CMC flow-path components to operate over 1000°F hotter than the state-of-the-art metallic concept adds system design flexibility to space-access vehicle concepts.

ACKNOWLEDGMENT

I wish to thank NASA, and in particular R. Baurle from NASA Langley Research Center for providing the VULCAN code to perform this analysis.

References

[1] Bogdanoff, D. W., "Advanced Injection and Mixing Techniques for Scramjet Combustors," Journal of Propulsion and Power, Vol. 10, No. 2, 1994, pp. 183-189.

[2] Drummond, J.P., "Enhancement of Mixing and Reaction in High-Speed Combustor Flowfields," in Advanced Computation and Analysis of Combustion, Edited by G.D. Roy, S.M. Frolov, and P. Givi, ENAS Publishers, 1997, pp.1-4.

[3] Sato, N., Imamura, R., Shiba, S., Takahashi, M, Tsue, M., and Kono, M., "Advanced Mixing Control in Supersonic Airstream with a Wall-Mounted Cavity," AIAA Paper 96-4510, 1996.

[4] Hartfield, Roy J., et. al., "Experimental Investigation of a Supersonic Swept Ramp Injector Using Laser-Induced Iodine Fluorescence," Journal of Propulsion and Power, Vol. 10, No. 1, 1994, pp. 129-135.

[5] Narayanan, A.K. and Damodaran, K.A., "Experimental Studies on Mixing of Two Co-Axial High-Speed Streams," Journal of Propulsion and Power, Vol. 10, No. 1, 1994, pp. 62-68.

[6] Rubins, P.M., "Review of Shock-Induced Supersonic Combustion Research and Hypersonic Applications," Journal of Propulsion and Power, Vol. 10, No. 5, 1994, pp. 593-601.

[7] Kumar, A., et. al., "Mixing Augmentation Technique for Hypervelocity Scramjets," Journal of Propulsion and Power, Sept.-Oct. 1989, pp. 514-522.

[8] Kraus, D.K. and Cutler, A.D., "Mixing of Swirling Jets in a Supersonic Duct Flow," Journal of Propulsion and Power, Vol. 12, No. 1, 1996, pp. 170-177.

[9] Zang, A.H., Yu, K., and Kothari, A., "Experimental Characterization of Mixing and Combustion in a Supersonic Duct," AIAA 2005-3303, AIAA/CIRA 13th International Space Planes and Hypersonic Systems and Technologies.

[10] Takahashi, M., Sunami, T., Tanno, H., Komuro, T., Kodera, M., and Itoh, K., “Performance Characteristics of a Scramjet Engine at Mach 10 to 15 Flight Condition,” Paper AIAA 2005-3350, AIAA/CIRA 13th International Space Planes and Hypersonic Systems and Technologies.

[11] Musielak, D.E., “Numerical Investigation of Fuel Injection, Mixing and Chemical Reaction for Application to Mach 10-12 Missiles,” A Technical Proposal to MSE Technology Applications, Inc., January 23, 2006.

[12] Young, G., Balar, R., Krasel, M. and Yu, K., “Effect of Nanoparticle Additives in Airbreathing Combustion,” AIAA 2006-7953.

[13] Huber, P.W., Schexnayder, C.J., and McClinton, C.R., “Criteria for Self-Ignition of Supersonic Hydrogen-Air Mixtures,” NASA TP 1457, 1979.

[14] Ben-Yakar, A., and Hanson, R.K., “Experimental Investigation of Flame-Holding Capability of a Transverse Hydrogen Jet in Supersonic Cross Flow,” Proceedings of the Twenty-Seventh International Symposium on Combustion, Combustion Institute, Pittsburg, PA, 1998, pp. 2173-2180.

[15] Musielak, D.E., “Numerical Simulation of Hydrogen Jet Interaction with Mach 4+ Airflow in A Scramjet Combustion Chamber.” Scramjet Development Program: Contractor Progress Report No. 1, February 2006.

[16] Musielak, D.E. and Micheletti, D., “Numerical Simulation of Cavity Flame Holders for Combustion and Flame Stabilization in a Mach 10-12 Scramjet Vehicle.” 2006 JANNAF CS/APS/PSHS Joint Meeting, San Diego, CA, December 4-8, 2006.

[17] White, J.A. and Morrison, J.H., “A Pseudo-Temporal Multi-Grid Relaxation Scheme for Solving the Parabolized Navier-Stokes Equations,” AIAA Paper No. 99-3360, June 1999. http://vulcan-cfd.larc.nasa.gov/WebPage/aiaa99-3360.pdf

[18] Musielak, D.E., “Numerical Simulation of Multiple Hydrogen Jet Flow Interaction in 2-D and 3-D Combustor Configurations.” Scramjet Development Program: Contractor Progress Report No. 3, April 2006.

[19] Kutschenreuter, P., “Supersonic Flow Combustors,” in Scramjet Propulsion, Editors Curran, E.T. and S.N.B. Murthy, Progress in Astronautics and Aeronautics, Vol. 189 (2000).

[20] Heiser W.H., and Pratt, D.T., Hypersonic Airbreathing Propulsion, AIAA Education Series, 1994.

[21] Gruber, M.R. (Wright-Patterson Air Force Base), Personal Communication, October 2006.

[22] Gruber, M.R., Nejad, A.S., and Dutton, J.C., “An Experimental Investigation of Transverse Injection from Circular and Elliptical Nozzles into Supersonic Crossflow.” Wright-Patterson Air Force Base, Tehnical Report WL-TR-96-2102.

[23] Lee, S.H., “Characteristics of Dual Transverse Injection in Scramjet Combustor, Part 1: Mixing.” J. of Propulsion and Power, Vol. 22, No. 5, Sept-Oct. 2006.

[24] Musielak, D.E., Micheletti, D. and Lofftus, D., “Numerical Simulation of Fuel Injection for Application to Mach 10-12 Vehicles.” AIAA Paper 2006-7902. 14th AIAA/AHI Space Planes and Hypersonic Systems and Technologies Conference, 6 - 9 Nov 2006.

[25] Musielak, D.E. and Micheletti, D., “Numerical Simulation of Cavity Flame Holders for Combustion and Flame Stabilization in a Mach 10-12 Scramjet Vehicle.” Presented at the 2006 JANNAF CS/APS/PSHS Joint Meeting, San Diego, CA, December 4-8, 2006.