EFFECTS OF FUEL INJECTION ON MIXING AND UPSTREAM INTERACTIONS IN SUPERSONIC FLOW

By

QIUYA TU

A DISSERTATION PRESENTED TO THE GRADUATE SCHOOL OF THE UNIVERSITY OF FLORIDA IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY

UNIVERSITY OF FLORIDA

2013
To my family
ACKNOWLEDGMENTS

I would like to sincerely thank my advisor Dr. Corin Segal for his guidance towards my study and research. Without his help and support, this work would not have been finished. I would also like to thank my family who are always my back when I went through the hard time. I would also like to thank my colleagues in the Combustion and Fluid Dynamics lab for their kindness and help.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>ACKNOWLEDGMENTS</th>
<th>.................................................................</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF TABLES</td>
<td>.................................................................................................</td>
<td>7</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>.................................................................................................</td>
<td>8</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS</td>
<td>..................................................................................</td>
<td>10</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>.................................................................................................</td>
<td>12</td>
</tr>
<tr>
<td>CHAPTER 1</td>
<td>INTRODUCTION .................................................................................</td>
<td>14</td>
</tr>
<tr>
<td>Air Breathing Engine</td>
<td>.........................................................................................</td>
<td>14</td>
</tr>
<tr>
<td>Turbojet Engine</td>
<td>.................................................................................................</td>
<td>15</td>
</tr>
<tr>
<td>Ramjet Engine</td>
<td>.................................................................................................</td>
<td>16</td>
</tr>
<tr>
<td>Scramjet Engine</td>
<td>.................................................................................................</td>
<td>17</td>
</tr>
<tr>
<td>Isolator / Combustor Aerodynamics during Transients</td>
<td>..........................................................</td>
<td>20</td>
</tr>
<tr>
<td>Optical Diagnostics for Pylon-Aided Fuel Injection on Mixing in Supersonic Flow</td>
<td>..........</td>
<td>23</td>
</tr>
<tr>
<td>Mixing Enhancement Strategies</td>
<td>...........................................................................</td>
<td>23</td>
</tr>
<tr>
<td>Planar Laser-Induced Fluorescence (PLIF)</td>
<td>..........................................................</td>
<td>26</td>
</tr>
<tr>
<td>Scope of Study</td>
<td>.................................................................................................</td>
<td>28</td>
</tr>
<tr>
<td>CHAPTER 2</td>
<td>EXPERIMENTAL SETUP ........................................................................</td>
<td>33</td>
</tr>
<tr>
<td>Scramjet Facility</td>
<td>.................................................................................................</td>
<td>33</td>
</tr>
<tr>
<td>Schlieren Photography Facility</td>
<td>..................................................................................</td>
<td>35</td>
</tr>
<tr>
<td>Optical Diagnostics</td>
<td>......................................................................................</td>
<td>35</td>
</tr>
<tr>
<td>Pylon Injector and Experimental Conditions</td>
<td>.........................................................................</td>
<td>35</td>
</tr>
<tr>
<td>Acetone PLIF Measurement System</td>
<td>........................................................................</td>
<td>35</td>
</tr>
<tr>
<td>Image Acquisition and Processing</td>
<td>........................................................................</td>
<td>37</td>
</tr>
<tr>
<td>CHAPTER 3</td>
<td>ISOLATOR / COMBUSTION CHAMBER INTERACTIONS ........................................................................</td>
<td>43</td>
</tr>
<tr>
<td>Experimental Results</td>
<td>.........................................................................................</td>
<td>43</td>
</tr>
<tr>
<td>Mach 1.6</td>
<td>.................................................................................................</td>
<td>43</td>
</tr>
<tr>
<td>Mach 1.9</td>
<td>.................................................................................................</td>
<td>45</td>
</tr>
<tr>
<td>Mach 2.5</td>
<td>.................................................................................................</td>
<td>46</td>
</tr>
<tr>
<td>Predicted vs. Measured Pressure Data</td>
<td>........................................................................</td>
<td>46</td>
</tr>
<tr>
<td>CHAPTER 4</td>
<td>OPTICAL DIAGNOSTICS FOR PYLON-AIDED FUEL INJECTION ON MIXING .............................................</td>
<td>57</td>
</tr>
<tr>
<td>Schlieren Photograph and Pressure Distributions</td>
<td>........................................................................</td>
<td>57</td>
</tr>
</tbody>
</table>
PLIF Results ........................................................................................................................................... 58
  Instantaneous Structures ....................................................................................................................... 58
  Ensemble-Averaged Structures ............................................................................................................ 59
  Effect of injection pressure ................................................................................................................. 59
  Effect of molecular weight .................................................................................................................. 61
  Geometrical features of jet plume ........................................................................................................ 61

5 ISOLATOR LENGTH DERIVATION ................................................................................................. 75
  Isolator Length Derivation without Mass Source Addition .............................................................. 77
  Isolator Length Derivation with Mass Source Addition ................................................................. 81

6 FLUENT SIMULATION: SHOCK WAVE/ BOUNDARY-LAYER INTERACTION ............................ 85

7 CONCLUSIONS AND FUTURE WORK ......................................................................................... 93
  Isolator / Combustion Chamber Interactions .................................................................................... 93
  Non-Reacting Mixing .......................................................................................................................... 94
  Isolator Length Derivation ................................................................................................................ 94
  Shock Wave/Boundary-Layer Interaction Simulation .......................................................................... 95
  Future Work ....................................................................................................................................... 95

APPENDIX

A DATA ACQUISITION AND CONTROL HARDWARE AND NATIONAL INSTRUMENTS HARDWARE ................................................................................................................. 97

B WALL FRICTION COEFFICIENT CALCULATION FORMULA ..................................................... 101

C SCRAMJET ACTUATOR ....................................................................................................................... 102

D PROGRAM PROCEDURE FOR PLIF IMAGES PROCESSING .................................................... 115

LIST OF REFERENCES .......................................................................................................................... 116

BIOGRAPHICAL SKETCH .................................................................................................................... 124
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Instrumentation specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>A-1</td>
<td></td>
</tr>
</tbody>
</table>

................................................................. 99
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-1</td>
<td>Specific impulse levels for several air breathing cycles and rocket propulsion...</td>
</tr>
<tr>
<td>1-2</td>
<td>Schematic diagram of a two-dimensional turbojet engine</td>
</tr>
<tr>
<td>1-3</td>
<td>Schematic diagram of a two-dimensional or planar geometry ramjet engine</td>
</tr>
<tr>
<td>1-4</td>
<td>Schematic diagram of a two-dimensional or planar geometry scramjet engine</td>
</tr>
<tr>
<td>1-5</td>
<td>Flow model for separation in a constant cross-section duct</td>
</tr>
<tr>
<td>1-6</td>
<td>Differential element of separated flow</td>
</tr>
<tr>
<td>1-7</td>
<td>Differential element of attached flow in a duct</td>
</tr>
<tr>
<td>2-1</td>
<td>Supersonic combustion facility</td>
</tr>
<tr>
<td>2-2</td>
<td>Schlieren photography schematic</td>
</tr>
<tr>
<td>2-3</td>
<td>Isolator and combustion chamber schematic and pylon geometry</td>
</tr>
<tr>
<td>2-4</td>
<td>Schematic of acetone PLIF measurement system</td>
</tr>
<tr>
<td>2-5</td>
<td>Location of the selected flow planes</td>
</tr>
<tr>
<td>3-1</td>
<td>Mach 1.6</td>
</tr>
<tr>
<td>3-2</td>
<td>Mach 1.9</td>
</tr>
<tr>
<td>3-3</td>
<td>Mach 2.5</td>
</tr>
<tr>
<td>3-4</td>
<td>Nozzle exit boundary layer</td>
</tr>
<tr>
<td>3-5</td>
<td>Predicted vs. measured pressure data</td>
</tr>
<tr>
<td>4-1</td>
<td>Schlieren photograph of the air flow without injection or pylon</td>
</tr>
<tr>
<td>4-2</td>
<td>Normalized pressure distribution at two different stagnation injection pressures</td>
</tr>
<tr>
<td>4-3</td>
<td>Instantaneous end-view PLIF images in three measurement planes for four different injection cases: plane 1 (left), plane 2 (center), and plane 3 (right)</td>
</tr>
<tr>
<td>4-8</td>
<td>Plume area comparison for each four cases along the streamwise direction for A) He and B) Ar</td>
</tr>
</tbody>
</table>
Plume penetration comparison for each four cases along the streamwise direction for A) He and B) Ar .......................................................... 73

Plume spreading comparison for each four cases along the streamwise direction for A) He and B) Ar .......................................................... 74

Overview of the density-based solution method ........................................ 89

2-D geometry model used in Fluent computation ........................................ 89

Mesh distribution in the model .................................................................. 90

Inlet total pressure profile after interpolation ........................................ 90

Computing procedure of Mach number contour for M=1.9 at 60% blockage, using $k$-$\varepsilon$ RNG model with enhanced wall treatment .............................. 91

Computing procedure of Mach number contour for M=1.9 at 60% blockage, using $k$-$\varepsilon$ RNG model with standard wall function .............................. 92

Data acquisition and control hardware ...................................................... 97

Trapezoidal motion profile of the linear actuator ...................................... 102

Sequence of control for the motion of the actuator arm ............................ 103
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>geometric area or spontaneous emission rate ($s^{-1}$)</td>
</tr>
<tr>
<td>$A_1$</td>
<td>isolator entrance area</td>
</tr>
<tr>
<td>$A_c$</td>
<td>the core flow area</td>
</tr>
<tr>
<td>$c$</td>
<td>speed of light</td>
</tr>
<tr>
<td>$C_{fr}$</td>
<td>wall friction coefficient at the separation point</td>
</tr>
<tr>
<td>$D$</td>
<td>diameter</td>
</tr>
<tr>
<td>$D_H$</td>
<td>hydraulic diameter</td>
</tr>
<tr>
<td>$dV_c$</td>
<td>collection volume ($cm^3$)</td>
</tr>
<tr>
<td>$E$</td>
<td>laser fluence ($J/cm^2$)</td>
</tr>
<tr>
<td>$F$</td>
<td>stream thrust</td>
</tr>
<tr>
<td>$F_1$</td>
<td>isolator entrance stream thrust</td>
</tr>
<tr>
<td>$f$</td>
<td>focal length</td>
</tr>
<tr>
<td>$f_1$</td>
<td>variable parameter</td>
</tr>
<tr>
<td>$g_1$</td>
<td>variable parameter</td>
</tr>
<tr>
<td>$H$</td>
<td>step height</td>
</tr>
<tr>
<td>$H_0$</td>
<td>stagnation enthalpy at the duct entrance</td>
</tr>
<tr>
<td>$h$</td>
<td>Prandtl constant</td>
</tr>
<tr>
<td>$K$</td>
<td>constant</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$M_1$</td>
<td>isolator entrance Mach number</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>$P_1$</td>
<td>static pressure at the duct entrance</td>
</tr>
<tr>
<td>$P_{in}$</td>
<td>static pressure at the duct entrance</td>
</tr>
<tr>
<td>$P_{out}$</td>
<td>static pressure at the duct exit</td>
</tr>
</tbody>
</table>
\( P_{oj} \)  
stagnation injection pressure

\( P_r \)  
pressure ratio

\( P_s \)  
static pressure at the isolator entrance

\( Q \)  
collisional quenching rate (s\(^{-1}\))

\( R_\theta \)  
Reynolds number

\( S \)  
intensity of the signal

\( \dot{S}_1 \)  
mass source term

\( \dot{S}_2 \)  
momentum source term

\( \dot{S}_3 \)  
energy source term

\( T \)  
static temperature

\( T_1 \)  
stagnation temperature

\( T_{0_{air}} \)  
air stagnation temperature

\( V \)  
volume or velocity

\( x \)  
distance or coordinate axis

\( y \)  
coordinate axis

\( z \)  
coordinate axis

\( \theta \)  
boundary layer momentum

\( \gamma \)  
constant, 1.4

\( \rho \)  
air density

\( \chi \)  
mole fraction of absorbing molecule

\( \varphi \)  
fraction of absorbed photons re-emitted as fluorescence

\( \sigma_{abs} \)  
molecular absorption cross section (cm\(^2\))

\( \eta_{optics} \)  
efficiency of the collection optics
Abstract of Dissertation Presented to the Graduate School of the University of Florida in Partial Fulfillment of the Requirements for the Degree of Doctor of Philosophy

EFFECTS OF FUEL INJECTION ON MIXING AND UPSTREAM INTERACTIONS IN SUPERSONIC FLOW

By

Qiuya Tu

December 2013

Chair: Corin Segal
Major: Mechanical Engineering

Scramjet engine performance has been studied experimentally and computationally almost under steady-state conditions. Transients of the airflow and fueling in the scramjet’s isolator or combustor create important fluid-dynamic/combustion interactions.

Spark schlieren photography was employed to study the effects of pressure rise in the combustion chamber on the isolator flow at three conditions with isolator entrance Mach number of 1.6, 1.9 and 2.5, covering the range of dual-mode combustion and transition to full scramjet operation. Heat release through combustion in the model scramjet was simulated by incrementally blocking the flow exit until upstream-interaction was induced and a shock train formed in the isolator. Theoretical predictions of the pressure rise in the isolator under separated flow conditions were calculated, which agreed well with the experimental data. The prediction is sensitive to the accurate modeling of the isolator inlet conditions and the correct selection of wall friction coefficient.

Gaseous helium and argon have been transversely injected into a Mach 1.6 airflow simulating a light and a heavy fuel injection behind a thin triangular pylon placed
upstream, in the isolator, which has a negligible impact on pressure losses. Planar laser-induced fluorescence (PLIF) was used to observe the penetration and mixing in the test section at three cross-sections including the recirculation region and beyond. Results were compared to the no-pylon cases, which showed the presence of the pylon resulted in improving both penetration and spreading of the jet.

Simulation for shock wave/ boundary-layer interaction was conducted in Fluent for case of $M=1.9$ at 60% blockage by using $k>-\varepsilon$ RNG model with two different near wall treatments. In both cases, the shock ran out of isolator before the computation converged, this is different from experimental results. Proper actual wall friction force may have a very important effect on the computation, which needs to be evaluated.
CHAPTER 1
INTRODUCTION

At present, transporting between the earth and space, rocket-based launch vehicles are the only tools, but their access to space is limited by their high costs with low useful payload, since they carry fuel and oxidizer along with them. However, airbreathing vehicles can resolve this problem by using oxygen from the atmosphere to propel themselves instead of carrying it onboard which would increase the useful payload weight and reduce launch costs.

Smart\(^1\) mentioned the important differences between rockets and airbreathing engines described by Antonio Ferri (Ferri 1964) as: the airbreathing engine’s potential specific impulse is much larger because it only takes fuels, its structural weight is larger because it must process air and has an intake, its thrust is a function of flight Mach number and altitude, it has severe structural problems associated with aerodynamic heating and vehicle drag introduced by flying in the atmosphere. So using hypersonic air breathing vehicles for launching payloads in space is a big interest in the world and it has been realized that a hypersonic airbreathing propulsion system could fulfill many roles that a rocket could not, like long-range hypersonic cruise and reusable space launch vehicles for efficient space access.

Air Breathing Engine

Propulsion for airbreathing launch vehicle would involve combined-cycle engines that utilize different forms of air breathing engines most suited for different stages of the flight envelope.

Figure 1-1 shows the estimated specific impulse of several air breathing cycles and rocket propulsion as the flight Mach number increases with operation of
hydrocarbon and hydrogen fuels\textsuperscript{2}. In the image, the specific impulse of all the air breathing engines (turbojet, ramjet, and scramjet) exceeds that of the rocket engine. Thus, air breathing propulsion is usually preferred over rocket propulsion when feasible air breathing technology exists that is suitable for the required flight regime. Around Mach number 6, ramjet is switched to scramjet cycles. As is also shown, the rocket thrust is almost constant regardless of the vehicle velocity, so it can achieve high thrust at takeoff; furthermore, air breathing engines are lack of turbo-machinery, which are incapable of producing thrust at takeoff. Hence, a combination of air breathing and rocket propulsion is often employed to take advantage of their relative strengths or compensate for their inherent weakness, which means rockets needs to be considered when air breathing engines cannot provide any thrust, like ramjets are standing still or scramjets are flown out of the atmosphere.

**Turbojet Engine**

Turbojet engine, also called jet engine, is a simple type of gas turbine engine, the oldest kind of general-purpose air breathing engine, is commonly used for high speed, high altitude, and air consumption engine. Figure 1-2 shows the schematic diagram of a turbojet engine. It consists of an air intake/inlet, an air compressor, a combustion chamber, a gas turbine and a nozzle. Subsonic air entering the engine inlet is slowed down to low subsonic speed by the diffuser, compressed to a high pressure and enters into the combustion chamber where heat is added to it by burning the fuel, then the high pressure, high temperature air is expanded out through the turbine into the nozzle where it is accelerated to high speed to provide propulsion\textsuperscript{3,4,5}. The work done on the turbine is used to drive the compressor.
Compared with other turbine engines, Husain\textsuperscript{5} summarized the turbojet engine as: low thrust at low speeds; relatively less complicated than other propulsion engines; long take off roll required; low specific weight per unit thrust; best suited for high attitudes, high speed and long distance travel; smaller frontal area less drag.

The turbojet engine can increase to supersonic speed about Mach number 2.5, which approaches its operational limit. A ramjet engine is suitable for higher Mach numbers.

\textbf{Ramjet Engine}

Ramjet is a form of airbreathing jet engine using the engine’s forward motion to compress incoming air without a rotary compressor, it does not need to be axially symmetric because it takes advantage of the outside surface of the vehicle functioning as the inside surface or the boundary of the engine\textsuperscript{6,7}. The two-dimensional geometry of ramjet engine is shown in Figure 1-3.

Heiser and Pratt\textsuperscript{6} gave a very detailed description of how ramjet engine works. When air enters the engine, it is compressed by reducing the speed through several steps, which include supersonic air passes through one or more oblique shock waves generated by the forebody of the vehicle or of the diffuser, through a convergent duct for further deceleration, then the supersonic flow passes through a normal shock wave system, transforming into subsonic flow, enters the combustor where the complex combustion takes place and the flow speed is further reduced. Through the convergent-divergent nozzle, the hot, high pressure flow from the combustor accelerates back to a certain supersonic exit speed and finally exhausts into the atmosphere. In this process, the thrust is generated by the flow passing through the ramjet because the high
temperature exhaust flow has more velocity and momentum leaving than it did entering. This reaction force is known as the internal or uninstalled thrust of the engine.

Although ramjets are the engines of choice for flight in the Mach number range 3-6, they cannot produce thrust when their speed is zero, some other propulsion system must be used to accelerate the vehicles to a speed where the ramjets begin to produce thrust.

Ramjet works efficiently by decelerating the supersonic air flow to subsonic, and is most efficient at Mach 3, but above Mach 5, ramjet propulsion becomes very inefficient due to dissociation and pressure loss caused by shock as the incoming air is slowed to subsonic velocities for combustion. The new supersonic combustion ramjet, or scramjet, solves this problem by performing the combustion supersonically in the combustor.

**Scramjet Engine**

A scramjet is a variant of a ramjet airbreathing jet engine in which combustion takes place in supersonic airflow. In ramjet, combustion of fuel with air takes place after the flow has been slowed internally to subsonic speeds. But in scramjet, airflow is supersonic throughout the entire engine. It is not easy to decelerate the flow to subsonic and even it can be done, it is not profitable. So, combustion must take place at locally supersonic conditions. Engines that operate in this way are known as supersonic combustion ramjets, or scramjets for short.

In the 1950s and 1960s a variety of experimental scramjet engines were built and ground tested in the US, Russia, France, Germany, Australia and Japan. A great deal of progress had been made to determine the engine’s characteristics, like internal
performance, component behavior, operating limits, heat transfer, durability and many other factors.

The flagship of the developmental scramjet engines was the Hypersonic Ramjet Experiment or Hypersonic Research Engine (HRE), a project funded by the National Aeronautics and Space Administration (NASA) and carried out largely by the Garrett Corporation from May 1964 to April 1975.

The first successful flight test of a scramjet was performed by Russia in 1991, the Russian Central Institute of Aviation Motors (CIAM) designed and launched a subscale model of a combined or “dual-mode” ramjet and scramjet airbreathing engine. At flight Mach number about 3.5, the combustion started in the ramjet mode, and at Mach number 5.0, the combustion took place in the transition to scramjet or supersonic, which showed ramjet to scramjet transition under realistic conditions. Then from 1992 to 1998 CIAM collaborated with France and then with NASA, USA and conducted an additional 6 flight tests of the axisymmetric high speed scramjet-demonstrator.

Maximum flight velocity greater than Mach 6.4 was achieved and scramjet operation during 77 seconds was demonstrated.

The history of scramjet programs through the world, up to year 2000, is well described in an article by Curran (2001).

Most recently, in March 2010, a (HIFiRE) hypersonic rocket was successfully tested by Australian and American scientists, it reached an atmospheric velocity of “more than 5000 kilometers per hour” after taking off. In May 2010, NASA and the United State Air Force successfully flew the X-51A Waverider for approximately 200 seconds at Mach 5, setting a new world record hypersonic airspeed. In November
2010, Australian scientists successfully demonstrated that the high speed flow in a naturally non-burning scramjet engine can be ignited using a pulse laser source\textsuperscript{14}. In August 2012, a further X-51A Waverider test failed\textsuperscript{15}. In May 2013 an unmanned X-51A Waverider reached “4828 km/h” (Mach 5.1) during a three-minute flight under scramjet power\textsuperscript{16}.

Scramjet engine is designed to operate in the hypersonic flight regime, and relies on the combustion of fuel and an oxidizer to produce thrust, but it does not decelerate the flow to subsonic velocities. When the airflow is decelerated, the relative velocity and kinetic energy decrease, any missing kinetic energy will become as internal energy due to the energy conservation, due to this, the pressure, temperature, and density of the flow entering the combustor are considerately higher than in the freestream\textsuperscript{6}.

To avoid the normal shock wave system, the flow cannot be slowed down significantly by the engine inlet and diffuser, so the flow entering the combustor is supersonic.

The schematic image of a two-dimensional geometry scramjet engine is shown in Figure 1-4. The hypersonic Mach number airflow is slowed down to supersonic speed by oblique shocks at the inlet and then by the diffuser. The diffuser is convergent, and gaseous or liquid fuel is injected into the flow downstream of it to increase the residence time and enhance the mixing. An isolator is located before the combustor to prevent the upstream flow from adverse back pressure induced by the heat release in the combustor, after the combustor, the supersonic flow goes through the divergent nozzle to further accelerate the speed, and some of the acceleration can take place outside the confining duct by using the afterbody of the vehicle as a free expansion surface\textsuperscript{6}.
Although scramjet has been studied for a while, but its difficulties of designing are still not resolved, which are\textsuperscript{1,2,6}

- Short residence time for stable, efficient fuel air mixing and combustion in a supersonic flow for a combustor of reasonable size, especially for liquid fuels which need additional processes like liquid breakup and vaporization.
- Flame stability and holding in high speed flow.
- Thermal choking control and high heat loads and friction losses in hypersonic flows.
- Non-equilibrium nozzle flows and loss of energy due to incomplete combustion in the high speed flow.
- Developing the analytical tools that can provide reliable prediction of the actual behavior.
- A low-speed propulsion system needed to take off the vehicle and accelerate it to the scramjet take-over flight Mach number from 3.5 to 5.
- Providing the structural integrity which requires some fitness and many compromises for adequate performance at the upper and lower limits of the desired speed range and can be reusable.

\textbf{Isolator / Combustor Aerodynamics during Transients}

The isolator is a duct which, in general, plays no role other than protecting the inlet flow from adverse back pressure. In principle, the isolator adds weight, internal drag and heat loads on the engine structure and, therefore, its length must be limited to the minimum necessary required by operability constraints\textsuperscript{2}. The isolator can be viewed as an extension of the inlet where additional compression takes place but its purpose serves the distinct function of protecting the inlet flow from adverse effects of combustion pressure rise. Boundary layers are present in the isolator after having formed on the long vehicle forebody and inlets surfaces. The interaction between the boundary layer and adverse pressure gradient may result in flow separation and formation of shock waves. In the isolator section of the duct the Mach number reduces
and the pressure and temperature increase due to the action of friction on the duct surfaces.

The flow field in the isolator shown in Figure 1-5 represents the case when compression is achieved both through an oblique shock train in supersonic region 1 and through area expansion of the subsonic flow in region 2. Near the wall, in region 3 a separation bubble forms to balance the pressure gradient across the isolator length through the shear stress. A repeated shock structure follows in the core of the duct. This flow structure established the combustion chamber entrance conditions where it further interacts with the heat release effects determining the flow stability and the process efficiency. The oblique shock train shown in Figure 1-5 is characteristic for a high Mach number operation. For moderate Mach number at the isolator entrance or a relatively thick boundary layer, a weaker train of oblique shocks take place in the form of a set of bifurcated shocks with the subsonic region 2 achieving a gradual compression in the isolator. In both cases, the shock train must ensure that the initial wave does not propagate upstream into the inlet to disrupt the flow, or, in the limit, result in the inlet unstart.

As a result of the formation of the shock train structure, the issue of immediate interest is the prediction of the minimal isolator length that can be allowed; a second issue of great importance is the losses related to the isolator drag. Once the isolator is sized, a critical issue is the shock train stability as the engine undergoes transients.

The isolator length can be determined, at nominal design conditions, by the pressure rise that must be achieved between the inlet outlet and the combustion chamber entrance and conservation equations in the separated flow. The differential
element of the separated duct is shown in Figure 1-6\(^1\), the core flow area \((A_c)\) is less than the geometric area \((A)\) when the flow is attached, as shown in Figure 1-7.

Based on a large amount of experimental data at different Mach numbers, Reynolds numbers and duct geometries, the pressure ratio \(P/P_i\) over a length \(dx\) was determined by Ortwerth (2001) to vary as\(^{17}\):

\[
\frac{d(P/P_i)}{dx} = 4K \gamma (P/P_i) M^2
\]  

Where \(4K = 44.5C_{fl}\) and \(C_{fl}\) is the friction coefficient at the initial separation point.

The isolator length, written in normalized parameters, depends on the pressure ratio, \(P_r = P_{in}/P_{out}\), as follows:

\[
\frac{X}{D_H} = \frac{1}{4K \gamma f_1} \left[ \frac{p_r - 1}{(f_1 - p_r)(f_1 - 1)} + \frac{1}{f_1} \ln \frac{p_r (f_1 - 1)}{(f_1 - p_r)} \right] + \frac{\gamma - 1}{2 \gamma} \ln p_r
\]  

Where

- \(4K\) is the constant friction coefficient at the duct entrance, taken as 44.5,
- \(D_H\) is the hydraulic diameter, \(4 \times \text{area/duct perimeter},\)
- \(f_1 = \frac{F}{P_i A}\), where \(F\) is the stream thrust, \(A\) is the area and the subscript refers to the isolator entrance station,
- \(g_1 = \sqrt{\frac{(\gamma - 1)H_0}{P_i A}}\), where \(H_0\) is the stagnation enthalpy at the duct entrance.

Equation 1.2 was found to predict the shock train length accuracy with 20% over a broad range of experimental results, including ducts of various shapes (round and rectangular), entrance Mach numbers ranging from 1.5-5, order of magnitude variation in the entrance Reynolds number, and different friction coefficients.
An addition correlation was offered by Waltrup and Billig\textsuperscript{19} based on experimental data for a circular duct, as shown in the following equation,

$$\frac{X}{\sqrt{D}} = \frac{\theta^{\frac{1}{2}}}{Re_{\theta}^{\frac{3}{4}}} \frac{1}{M_1^2 - 1} \left[ 50(p_r - 1) + 170(p_r - 1)^2 \right]$$  \hspace{1cm} (1.3)

which emphasizes the dependence of the shock train length on the Reynolds number, $Re_{\theta}$, the boundary layer momentum thickness, $\theta$, and the isolator entrance Mach number, $M_1$. Eq. 1.3 indicates that, for a fixed pressure ratio, the shock train length increases for flows with shock boundary layers at the isolator entrance and decreases with increased Reynolds and Mach numbers. Thicker boundary layers separate under weaker shock waves and a smaller angle of the initial shock wave results in a long shock train. As the Reynolds number increases, the boundary layer can withstand stronger shocks and the angle of the initial shock that generates the separation is more abrupt resulting in a shorter shock train. The Mach number effect is not immediately evident from the equation because it is coupled to the pressure ratio. As the Mach number increases, the required compression that would lead to separation is higher. The angle of the shock wave that leads to separation is lower and the shock train is longer.

**Optical Diagnostics for Pylon-Aided Fuel Injection on Mixing in Supersonic Flow**

**Mixing Enhancement Strategies**

The short residence time in practical supersonic combustion systems, typically of the order of a few milliseconds imposes severe requirement for mixing – and vaporization if liquid fuels are used – to ensure efficient heat release and positive net thrust generation\textsuperscript{2}. This causes the mixing limited in combustion in supersonic flows\textsuperscript{20,21}. The issue of mixing enhancement is, therefore, of particular interest for these devices.
Various types of fuel injection configurations and injector shapes have been studied for mixing enhancement mostly focused on changing the flowfield within the combustor\textsuperscript{22,23}. Transverse fuel injection relative to the main combustor flow provides good penetration and mixing compared to parallel injection, but at the expense of shock losses\textsuperscript{24-27}. For low hypersonic flow, the increased penetration and mixing of transverse injection is favored. However, at higher hypersonic Mach numbers, the shock losses become too severe and parallel injection is favored. Attention has also been given to injection in the presence of solid\textsuperscript{28-32} or aerodynamic ramps\textsuperscript{33}. Straight or swept ramps that produce near parallel injection have shown reasonable far-field mixing\textsuperscript{34,35,36}, although their near-field mixing performance falls below transverse injection alternatives. The ramp vortex shedding provides a means to lift the fuel from a low injection angle and promotes penetration into the core air stream. Because physical inflow ramps require cooling, especially in localized hot spots such as in recirculation regions, the aerodynamic ramp\textsuperscript{37,38} or an angle-injection solution\textsuperscript{39,40} from a flush-wall have been suggested as non-cooled injection configurations.

A solution that takes advantage of the high penetration of transverse injectors without the penalty of high pressure losses are the pylon-based injectors suggested by Vinogradov and Prudnikov\textsuperscript{41}. It involves thin, swept pylons with the fuel injected transversely in the separated region behind them. The results showed that the penetration increase with these pylons was substantial. Livingston et al.\textsuperscript{42} showed that thin pylons can be used with minimal pressure losses and applied this type of injection in an inlet, upstream of the isolator to provide additional mixing length. Hence, it is possible to achieve considerable penetration with relatively low dynamic pressure ratios,
even less than unity, using this type of pylons. This is significant in particular when considering that in most cases normal injection from the wall requires dynamic pressure ratios of the order of $10^{-15}$. A review of thin pylons applications is given by Vinogradov et al.\textsuperscript{44}.

To increase the residence time and achieve a higher degree of mixing in the combustion chamber it may be useful to inject part of the fuel upstream, in the isolator, in the inlet or further upstream on the vehicle body. In this case a complex but more flexible system is obtained; the optimization of this system could result in multiple advantages including (i) mixing enhancement; (ii) shorter isolator and combustor, consequently, reduced weight and cooling loads; (iii) a more flexible fuel control system due to the possibility of distributing the fuel between the preinjection region and the combustor and (iv) the possibility of injecting combinations of liquid and gaseous fuels in different regions\textsuperscript{2,45}.

When the fuel is injected upstream, there is a danger of flashback due to fuel remaining in the boundary layer potentially causing upstream flame propagation. With the pylons described here penetration increases and the residual fuel in the boundary layer is avoided. Owens et al.\textsuperscript{45}, Shikhman et al.\textsuperscript{46}, Vinogradov et al.\textsuperscript{47} and Guoskov et al.\textsuperscript{48} showed in combustion experiments that fuel injection upstream of the combustion chamber was possible without flashback. The same is true for liquid-fuel injection as shown by the experiments by Livingston et al.\textsuperscript{42} where, in an inlet operating at $M = 3.5$, the pylon helped to remove the fuel entirely away from the wall. Most significantly, from the mixing enhancement point of view, the jet experienced an abrupt breakup and was
carried into the inlet core airflow at the pylon height. Hence, the pylon’s presence helped placing the fuel in a favorable mixing region.

More recently, Gruber et al.\textsuperscript{49} confirmed these results evaluating pylon-aided fuel injection with three pylon geometries. In all cases the presence of the pylon resulted in improved fuel penetration without leading to significant total-pressure-loss characteristics. Computationally, Pohlman and Greendyke\textsuperscript{50} obtained similar results using five triangular pylons.

**Planar Laser-Induced Fluorescence (PLIF)**

Laser diagnostics is a non-intrusive flow measurement tool which has been widely used in recent years. Planar laser-induced fluorescence, or PLIF, is such a popular technique. It can acquire planar snapshots of flow in various flow systems. Except as a flow visualization technique, PLIF can also be used to measure gas-dynamic properties such as velocity, temperature, pressure, mole fraction, gas density or species concentration in wide range of environments\textsuperscript{51-55}. Hanson et al.\textsuperscript{56}, Schulz and Sick\textsuperscript{57} provided an overview of these. PLIF images are obtained by using a laser to excite a molecular species present in a flow. This species either occurs naturally or can be seeded. Commonly used species include NO, iodine, OH, and acetone. The fluorescence signal $S_f$ (number of counts recorded on a pixel of a detector) can be expressed as follows.

$$S_f = \eta_{\text{optics}} \frac{E}{hc/\lambda} dV_c \frac{\chi P}{\kappa T} \sigma_{\text{abs}} \frac{A}{A + Q}$$ \hspace{1cm} (1.4)

Where

- $\eta_{\text{optics}}$ is the efficiency of the collection optics,
- $E$ is the laser fluence (J/cm$^2$),
- $hc/\lambda$ is the energy of a photon,
- $dV_c$ is the collection volume (cm$^3$),
- $\chi$ is the mole fraction of absorbing molecule,
- $\sigma_{abs}$ is the molecular absorption cross section (cm$^2$),
- $A$ is the spontaneous emission rate (s$^{-1}$),
- $Q$ is the collisional quenching rate (s$^{-1}$).

The term $\frac{A}{A+Q}$ is known as the fluorescence yield $\varphi$, the fraction of absorbed photons re-emitted as fluorescence photons. It depends on composition, temperature and pressure of the gas mixture, and excitation wavelength. $\sigma_{abs}$ depends on gas temperature and excitation wavelength. Hence the fluorescence signal is a function of species mole fraction, temperature, pressure and excitation wavelength.

Fox et al.$^{58}$ used NO PLIF to evaluate the fuel mole fraction distribution and the mixing performance of various fuel injectors in supersonic flow by images in streamwise and spanwise planes. McMillin et al.$^{59}$ employed NO PLIF to measure the temperature in a hot, nonoxidizing supersonic cross flow with fuel injected. Gruber et al.$^{49}$ applied NO PLIF to study the instantaneous jet plume structure when gaseous fuel was injected into a Mach 2 non-reacting crossflow with the aid of pylon.

Fletcher et al.$^{60}$, Hartfield et al.$^{61}$, used I$_2$ PLIF to measure the steady, non-reacting compressible flow quantities, density, pressure, temperature and velocity. Hartfield et al.$^{62}$, Abbitt et al.$^{63}$, Hollo et al.$^{64}$ employed I$_2$ PLIF to measure injectant mole fraction distribution in non-reacting supersonic combustor and evaluate mixing rates for different injection configurations. Handa et al.$^{65}$ investigated the shock wave/ boundary layer interaction in a straight duct flow with I$_2$-PLIF.

Morris et al. employed OH-PLIF to investigate flowfields produced by a wedge mounted on a Ram-accelerator$^{66}$ and reactive flows$^{67}$. Vaidyanathan et al.$^{68}$ estimated
the uncertainties associated with OH PLIF measurement in an oxygen-hydrogen system produced by a shear coaxial injector. Singla et al.\textsuperscript{69} applies OH PLIF to study the high-pressure cryogenic flames structure in the injector near-field.

Lozano et al.\textsuperscript{70} explored the use of acetone (CH\textsubscript{3}COCH\textsubscript{3}) as a suitable tracer for PLIF concentration measurements in gaseous flows. Grossmann et al.\textsuperscript{71} studied the acetone fluorescence intensity as a function of temperature and pressure. Thurber et al.\textsuperscript{72} investigated the temperature and excitation wavelength dependence of acetone LIF signal. VanLerberghe et al.\textsuperscript{73} used acetone PLIF to investigate mixing of an underexpanded sonic jet injected transversely in a supersonic crossflow. Thakur\textsuperscript{74} used acetone PLIF to get a planar distribution of fuel mole fraction in the recirculation region formed by a rearward facing two-dimensional step.

**Scope of Study**

The focus of this work is to experimentally evaluate the isolator/combustion chamber interactions and the fuel distribution for flameholding analysis in the supersonic airflow. It not only provides direct insights into the design attributes that indicates supersonic combustor behavior, but can also be used to facilitate the design and optimization of an entire scramjet flowpath consisting of the inlet, isolator, combustor, and nozzle. Most of the existing studies and engine development data base are limited to steady operations. This work will thus considerably expand the current knowledge by considering airflow and fueling transients.

This study used schlieren photography method to show the effect of pressure rise in the combustion chamber on the isolator flow at three conditions with isolator entrance Mach number of 1.6, 1.9 and 2.5, which cover the range of dual-mode combustion and transition to full scramjet operation. Acetone PLIF was employed to
visualize the fuel mixing in non-reacting flow when the fuel was injected with the aid of pylon, in the direction normal to the airflow. Compared with the results without the aid of pylon, two factors, stagnation injection pressure and fuel molecular weight, were considered on the effect of penetration and mixing. Assume the flow in the isolator is one dimensional and in the separated region the pressure rise over length is linear with local dynamic pressure, the relations between isolator length and wall pressure rise were derived without mass addition and with mass addition. Certain cases of shock wave/ boundary layer interaction were computed by Fluent and compared with the experimental results.
Figure 1-1. Specific impulse levels for several air breathing cycles and rocket propulsion

Figure 1-2. Schematic diagram of a two-dimensional turbojet engine
Figure 1-3. Schematic diagram of a two-dimensional or planar geometry ramjet engine

Figure 1-4. Schematic diagram of a two-dimensional or planar geometry scramjet engine

Figure 1-5. Flow model for separation in a constant cross-section duct
Figure 1-6. Differential element of separated flow

Figure 1-7. Differential element of attached flow in a duct
CHAPTER 2
EXPERIMENTAL SETUP

This chapter includes detailed descriptions of the experimental test facilities, operating conditions and diagnostic methods.

Scramjet Facility

The supersonic combustion facility, shown in Figure 2-1 has been described in detail by Segal and Young\textsuperscript{75}. It provides tests with a variable combustion chamber entrance Mach number of 1.6 – 3.6 and stagnation temperatures corresponding to Mach 3.0 – 4.8 flights. The flight Mach numbers correspond to the transition phase from ramjet to scramjet engine. The wind tunnel is a continuously operating facility with 0.75 kg/s airflow which can be heated up to 1200 K by using a vitiated heater based on hydrogen combustion with oxygen replenishment, electronically controlled by a fuzzy logic controller\textsuperscript{76} to maintain a constant 0.21 oxygen mole fraction at all conditions, and to maintain the required constant stagnation temperature at the heater exit. All the experiments discussed here were performed with cold air $T_{0\text{air}} = 300$ K. The heated air is then directed via a bell mouth with compression on four sides to the interchangeable supersonic nozzle with compression on two sides, which can provide a range of isolator entrance Mach number from Mach 1.6 to 3.6. Each nozzle has a fixed $2.54 \times 2.54 \text{ cm}^2$ exit area. The nozzles are instrumented with a pressure port and transducer to measure the freestream static pressure at the entrance to the test section. A constant area, $2.54 \times 2.54 \text{ cm}^2$, isolator is placed between the nozzle and the test section to protect the nozzle from adverse backpressure which is generated by the combustion in the test section. Optical access is available to the isolator’s flow from two sides. The combustion chamber, with a constant cross-section area, has a rearward facing step with height $H =$
12.5 mm on two sides acting as a quasi two-dimensional flameholder, and has $26H$ in length. The test section is symmetric and has the option of optical access through quartz windows. It incorporates 15 pressure ports and 5 thermal couples and was water-cooled for combustion tests.

There are six fuel injection locations; three locations can be simultaneously and independently controlled with non-similar fuels, i.e., a combination of gas – gas or gas – liquid fuels. The purpose of multiple injection locations capability is to offer fuel modulation. For the liquid fuel injection, accurate measurement can be obtained by the calibrated flowmeters. In this study, gaseous fuel was injected transverse to the airflow at a distance $10H$ upstream of the step with a 1.0 mm diameter hole on the side of isolator wall. Helium - having molecular weight close to hydrogen, and argon - having molecular weight close to propane, were injected as simulated fuel in non-reacting flow test.

A LabView program, Data acquisition and control (DAQC) hardware and associated National Instruments hardware (shown in Appendix A) were used to monitor experimental conditions such as airflow stagnation pressure, temperature and Mach number at the nozzle exit, fuel stagnation pressure and temperature, test section wall static pressure and temperature distribution along the airflow direction. Schlieren photography was employed to study the effects of pressure rise in the combustion chamber on the isolator flow and planar laser induced fluorescence (PLIF) was used to observe the fuel penetration and mixing in the test section at three cross-sections including the recirculation region and beyond as described below.
Schlieren Photography Facility

The schematic of the spark schlieren photography system is shown in Figure 2-2. The light source is SensiFlash 1500 from the Cooke Corporation which needs to be charged for 2 minutes for each full spark. After the light passed through a pin hole with 1 mm diameter, it was reflected by a parabolic concave mirror with 2000 mm focal length through the windows of isolator or test section, another identical parabolic concave mirror on the other side of the scramjet facility reflected the light through a knife-edge whose direction was parallel to the main stream, then the upside down image was shown on the white screen, which can be captured by a Cooke corporation intensified digital charge-coupled device (ICCD) camera.

Optical Diagnostics

Pylon Injector and Experimental Conditions

Gaseous helium (He) and argon (Ar) were used as injectants to simulate a light fuel (hydrogen) and a heavy fuel, e.g., propane to evidence the effect of molecular weight. The fuel was transversely injected into the supersonic crossflow from a 1 mm diameter orifice located at $10H$ upstream of the step, in the isolator as shown in Figure 2-3. Two different stagnation injection pressures were applied: 2.4 atm and 5.1 atm. Both pylon and non-pylon configurations were evaluated. The pylon was designed as shown in Figure 2-3, to minimize the aerodynamic drag; hence, the thickness was selected as 2.3 times the injector diameter with swept leading edge and triangular cross section based on previous design recommendation.

Acetone PLIF Measurement System

Figure 2-4 illustrates schematically the acetone PLIF system used for measurements. The fourth harmonic from a Spectra-Physics Nd: YAG laser (GCR-150)
was used with a wavelength of 266 nm and output energy of 0.75 W at 10 Hz. The beam was expanded into a two-dimensional sheet of 50 mm wide and 0.5 mm thick by using a range of optics suited for UV range of laser light. The optical path included three 25.4 mm diameter mirrors, two cylindrical convex lenses of \( f = 100 \) mm and 500 mm, and one cylindrical concave lens \( f = -100 \) mm, so that the flowfield could be probed with a vertical laser sheet. The fluorescence images were recorded using a Cooke corporation intensified digital charge-coupled device (ICCD) camera and its associated software and a Sigma 50 mm/2.8 camera lens. The camera has a resolution of 1024×1280 pixels, 12-bit dynamic range and shutter speed down to 3 ns. The software was used to set the camera parameters before taking images and also to save the captured images on the computer. The camera gate was set to 10 ns to collect the acetone fluorescence’s life time of 4 ns. The devices were synchronized by a pulse generator. A band pass filter (335 – 610 nm) and a short pass filter (\(~500\) nm) were placed in front of the camera to eliminate elastic light reflections. The spatial resolution of the camera was 62.5, 104.2, and 63.3 \( \mu \)m/pixel for plane 1, 2, and 3, respectively. The injectant density change due to acetone seeding was estimated to be less than 1.4% assuming saturated condition at the injector. Therefore, this level of acetone seeding caused a negligible influence on the injectant density.

The intensity of laser-induced fluorescence from the acetone molecule depends on the local temperature, pressure, mole-fraction, and the coexisting species and the intensity of the signal \( S \) was translated into the acetone molar concentration (mol/m\(^3\))\(^77\). The error was estimated to be 6.5% when assuming a linear relationship by the method described in Thurber’s dissertation\(^78\). With the assumption that acetone distribution in
the flow is the same as that for the fuel, the fuel mole fraction distribution in the flow was determined quantitatively. Figure 2-5 shows the step and the location of three laser sheet planes. The injection was at $10H$ upstream of the step and the laser sheet planes were at $0.5H$, $2H$ and $10H$ downstream of the step, hence in the recirculation region close to the step, towards the end of the recirculation region - since the reattachment was at $2.7H^{79}$ downstream from the step— and further downstream in the far field.

**Image Acquisition and Processing**

After getting the images, they were processed by subtracting the background, correcting the laser sheet uniformity and normalizing the intensity. Background image was acquired from about 100 images taken just prior to runs of each day. From each image taken in run, the background image was subtracted to eliminate scattered noise and dark current noise, and the non-uniformity of the laser-sheet intensity distribution was corrected. In addition, the intensity range of each single shot image was normalized by the maximum value in each image. Owing to the normalization, the influences of the shot-by-shot and run-by-run fluctuation of acetone doping rate as well as the fluctuation of each laser pulse-intensity were canceled out.

For the check of laser-sheet uniformity, reference images of laser-sheet were taken with the duct filled with acetone-doped gas without any flows, i.e. the test section was closed and full of acetone vapor. Then we selected one line normal to the laser-sheet axis in the averaged reference image, and took the traveling average of the LIF-signal intensity on the line. Finally the distribution on the line was extended to whole LIF image. Fluctuation of each laser pulse is believed to be removed by taking an average. All the processes were finished by a Matlab program procedure attached in Appendix D.
The camera setting positions are schematically illustrated in Figure 2-4. For images of the y-z planes, or the cross sectional images, photographs were taken by a camera set to 45 degree against the cross section due to the physical restriction. The obliquely acquired images were corrected by use of the perspective transformation\(^8\) to extract end-view images in the post-processing.
Figure 2-1. Supersonic combustion facility

Figure 2-2. Schlieren photography schematic
Figure 2-3. Isolator and combustion chamber schematic and pylon geometry. The fuel was transversely injected into the supersonic crossflow from a 1mm-diameter orifice located at 10H upstream of the step, i.e. in the isolator. The pylon is 7.5 mm high and 2.3 mm wide at the base. Windows allow flow access for PLIF and visualization in the isolator and the test section.
Figure 2-4. Schematic of acetone PLIF measurement system. The laser beam was expanded into a two-dimensional sheet of 50 mm wide and 0.5 mm thick at three heights along the vertically oriented test section.
Figure 2-5. Location of the selected flow planes. The injection was at $10H$ upstream of the step and the laser sheet planes were at $0.5H, 2H$ and $10H$ downstream of the step. The $x$, $y$, $z$ axis correspond to the streamwise, transverse and spanwise direction. The three planes investigated are in the recirculation region close the step, towards the end of the recirculation region – since the reattachment was at $2.7H$ downstream from the step – and further downstream in the far field.
CHAPTER 3
ISOLATOR / COMBUSTION CHAMBER INTERACTIONS

The chapter experimentally studied the effects of pressure rise in the combustion chamber on the isolator flow at three conditions with isolator entrance Mach number of 1.6, 1.9 and 2.5 and $T_{0\text{air}}=300$ K. Spark schlieren photography and wall pressure measurements were employed, the configuration is described in Chapter 2. These conditions cover the range of dual-mode combustion and transition to full scramjet operation. Heat release through combustion in the model scramjet was simulated by incrementally blocking the flow exit until upstream-interaction was induced and a shock train formed in the isolator. Theoretical predictions of the pressure rise in the isolator under separated flow conditions agreed well with the experimental data. The prediction is sensitive to the accurate modeling of the isolator inlet conditions and the correct selection of wall friction coefficient.

**Experimental Results**

The pressure rise in the test section was obtained by physically blocking of the test section exit via a mechanism that can modulate the degree of blockage and the rate at which the exit is blocked thereby simulating fuel transients and the corresponding isolator/combustion chamber flow structure response. In this study the blockage was incremental until the isolator flow was completely affected with steady recordings at all levels. A vertical knife edge was used in the schlieren images shown here.

**Mach 1.6**

In this set of experiments, the shock train was propagating in an upstream direction as the simulated heat release increases through exit area blockage. Figure 3-1 shows the flow structure over the isolator and the test section with different blockage
levels indicated by the percentage of area reduction and the corresponding pressure rise. The isolator entrance Mach number is 1.6 and the flow is from left to right in the images. The isolator covers the first five pressure ports and the test section covers the following seven pressure ports. The wall pressure distribution shown in the figure is normalized by the static pressure at the nozzle exit and the axial distance is normalized by the step height. The origin is placed at the step location.

In the schlieren photographs, weak Mach waves can be seen in the incoming flow. These were generated by the growing boundary layer in the nozzle and by a slight mismatch at the nozzle/isolator interface. The pressure increases in the isolator is due to the boundary layer growth. In the absence of any blockage, i.e. no heat release, a sharp pressure drop is noted in the base of the step as the flow expands around the corner; then the pressure rises following reattachment at about $2.7H$ and, thereafter, it adjusts to the shock pattern in the combustion chamber.

As the simulated heat release increases through exit area blockage the pressure is increased, the shock system adjusts itself with the effects propagating in an upstream direction. The recirculation region increases substantially and, as seen in the 40% blockage, it occupies long regions along the combustion chamber walls flanking the core flow which remain supersonic. The supersonic jet emerging in the surrounding subsonic jet in the test section exhibits “side-to-side” fluctuations that can be attributed to a Coanda effect noticed in bounded flows. The isolator flow is still protected from these changes until the pressure rises to $P/P_s$ of 2.2 which is expected for the thermally chocked condition for this Mach number, i.e. $M=1.6$. 
Once the shock train has moved upstream into the isolator, as seen in the 66% blockage case, the typical pattern of shocks in a duct with moderate boundary layer thickness is observed. The first shock is bifurcated, while the subsequent shocks are not. Three parts of the bifurcated shock, leading oblique shock, nearly normal outer shock, and a trailing oblique shock, intersect at the bifurcation point. The outer normal shock is concave, facing upstream. A slip line is generated at the bifurcation point and extends downstream. The following shocks are similar but unbifurcated, the outer region is nearly normal, being concave facing downstream\textsuperscript{81,82,83}. It is clearly visible that within the single shock train in each photograph, the spacing between two successive shocks decreases. And aside from the weak waves that penetrate from the nozzle/isolator interface, the shock train in the isolator remains symmetric throughout the entire interaction.

The pressure ratio is exceeded as the “heat release” continues and the effects of flow changing in the isolator are noticed in the following figures. Above 65% blockage the characteristic shock structure is noticed with clear separation of the boundary layer along the isolator walls. The test section is almost entirely operating in a subsonic mode. The shocks can be pushed entirely upstream of the isolator. Any further heat release would result, in an actual engine, in upstream effects in the inlet with unstart.

**Mach 1.9**

With further increase in Mach number and for a significant upstream interaction, the point of bifurcation of the leading shock moves away from the wall and reaches the centerline of the duct. Then the normal part of the leading shock disappears. It generates an oblique shock\textsuperscript{18}. This is shown in the Mach 1.9 case in Figure 3-2. The Mach 1.9 case follows a similar trend to Mach 1.6 with the exception that the flow is
more resilient to effects of heat release, the number of shocks increases and the spacing between two consecutive shocks becomes much larger, i.e. the interaction region becomes longer. From the pressure plot, the thermal choking is experienced at Mach 1.9 for a pressure ratio rise close to 3. Further “heat release” generates oblique shock/normal shock train in the isolator duct following separation and establishment of a shock structure bounded by separated regions. The test section becomes entirely subsonic. Eventually, the upstream interaction penetrates the inlet where it can cause possible spillage and hammershock effects.  

**Mach 2.5**  
At Mach 2.5, shown in Figure 3-3, continuously increasing the blockage, i.e. with more heat release in the test section, the supersonic turbulent flow in the combustion chamber resided only along the center surrounded by thick subsonic regions. In the isolator, the bifurcated shocks were less apparent but the separation increased. Increasing the blockage to 72.5%, an asymmetric oblique shock was noticed in the isolator. The bottom leading oblique shock originated further upstream of the location where the top leading oblique shock originated. After intersecting, the shocks led to large separation regions starting from the origination points of leading shocks. A large amplification in the scale of the turbulent structure was observed going through the interaction and the boundary layer was seen to thicken substantially through the interaction.  

**Predicted vs. Measured Pressure Data**  
The prediction of pressure rise in the isolator given by Eq.1.2 results from the 1-D solution of the equations of motion. The solution applies to separated flows and the
computation starts at the location where the flow separates and it takes into account mass and momentum deficits at the isolator entrance.

The 63.5% blockage for Mach 1.9 case was used as an example, which shows the separation starts at the first pressure port, which is located at \( x/H = -15 \) in the facility layout. Figure 3-4 shows the nozzle exit boundary layer measurement, which is used to calculate the isolator entrance area deficit. Figure 3-5 shows the predicted vs. the measured pressure rise. Two predicted pressure rises are included in the figure: one with wall friction coefficient \( C_{fr} = 0.0109 \) which coincides with the experimental data well initially, then gradually increases with the error in the range of 10\%, and the other one with \( C_{fr} = 0.005 \) selection which underpredicts the pressure rise by as much as 36\%. This is a strong effect indicating the need to judiciously select the wall friction. Other factors should be included, such as, wall heat transfer, potential mass transfer, core pressure distortion, etc.
Figure 3-1. Mach 1.6. Selected conditions of blockage representing combustion induced pressure rise showing the upstream interaction penetrating the isolator above a pressure rise of 2.2 in the test section which corresponds to the thermal choking condition. A) 0% blockage, B) 40% blockage, C) 64.5% blockage, D) 65% blockage, E) 65.5% blockage, F) 66% blockage, G) 66.5% blockage, H) 67% blockage.
Figure 3-1. Continued
Figure 3-1. Continued
Figure 3-2. Mach 1.9. Isolator interaction is noticed when the pressure rises in the combustion chamber above 3. A) 0% blockage, B) 25% blockage, C) 30% blockage, D) 60% blockage, E) 61% blockage, F) 61.5% blockage, G) 62% blockage, H) 63% blockage, I) 63.5% blockage
Figure 3-2. Continued
Figure 3-2. Continued
Figure 3-2. Continued
Figure 3-3. Mach 2.5. Isolator interaction is noticed when the pressure rises in the combustion chamber above 5. A) 0% blockage, B) 50% blockage, C) 72.5% blockage
Figure 3-4. Nozzle exit boundary layer

Figure 3-5. Predicted vs. measured pressure data
CHAPTER 4
OPTICAL DIAGNOSTICS FOR PYLON-AIDED FUEL INJECTION ON MIXING

Previous analyses have shown that mixing can be enhanced using thin pylons that have only a negligible impact on pressure losses. In this study, helium and argon have been transversely injected into a Mach 1.6 non-reacting airflow simulating a light (hydrogen) and a heavy fuel (propane) injection behind a thin triangular pylon placed upstream, in the isolator at $T_{0\text{air}}=300$ K. It gives the fuel distribution just before ignition of fuel-air mixture. Penetration and mixing in the test section were observed at three cross-sections including the recirculation region and beyond with planar laser-induced fluorescence (PLIF), the configuration was described in Chapter 2. Results were compared to the no-pylon cases. The presence of the pylon resulted in improving both penetration and spreading of the jet and, at the same time, in lowering the concentration gradients in the recirculation region, an indication of improved flameholding ability.

**Schlieren Photograph and Pressure Distributions**

Figure 4-1 is a schlieren photograph of the air flow before fuel injection without the aid of pylon showing the isolator and the combustion chamber. There are weak Mach waves in the isolator due to a slight misalignment of the nozzle and isolator interface. In the combustor the air flow expands around the step and reattaches at $2.7H_{79}$. The recirculation region formed behind the step receives different amounts of fuel depending on presence of the fuel injection in the isolator in this case, or downstream as done in other configurations. The resulting composition has a critical effect in the flameholding ability$^{86}$.

The test section wall pressure distribution shown in Figure 4-2, (A) for He and (B) for Ar indicates that there is a pressure increase of 0.2 atm immediately behind the
pylon and no difference downstream of the step. It should be noted that the isolator pressure rise is local, behind and aligned axially with the pylon without effect in the rest of the flow indicating that the presence of the pylon causes essentially no pressure loss.

**PLIF Results**

Instantaneous and ensemble-averaged images described below provide details of the flow structure emphasizing the details of penetration and spreading, two main factors influencing fuel-air mixing.

**Instantaneous Structures**

Figure 4-3 presents representative instantaneous PLIF images taken at the selected 3 planes for four cases, which correspond to $x/H=0.5$, $2$ and $10$ downstream of the step, hence, the first two planes are in the recirculation region and the third is further downstream. In each image, the main flow direction is out of the paper plane, the axes are normalized by the step height, $H$. The origin is placed at the center of the duct in the $y$ direction and at the step in the axial direction, $x$. The injection location is in the isolator at $z/H=0$, $x/H=-10$, and $y/H=-1$. The LIF intensities are normalized by the maximum intensity in each plane. The highlighted black solid line shows the step.

The instantaneous images show to a certain extent of the turbulent structures which include both the vortical structures and the flow turbulence effects. In the near-field a compressibility effect is noticed due to molecular weight differences creating a difference in the structure size. But in the far-field the compressibility effect seems to weaken because almost no difference in structure sizes is noticed.

Without the pylon the fuel penetrates rapidly in the recirculation region through the shear layer but remains confined to a small region. With the pylon the instantaneous
structure is larger and it stretches in vertical direction while in non-pylon cases the plume occupies a smaller region indicating less penetration and spreading. The structure due to the shear\cite{87,88} effect is seen at the periphery of the jet plume. In some cases part of the plume is removed from the rest as seen in plane 3 as shown in Figure 4-3(C). Moreover, the jet plume often reaches the opposite wall for pylon-assisted cases. For the higher molecular weight injectant, i.e., argon, most of the injectant remains close to the injection wall, an effect of lower diffusion.

The instantaneous image results reveal the complex nature of the injectant/air interaction, which is principally responsible for mixing; they also indicate that highly intermittent and dynamic features still exist in the far-field. The main effect of the pylon is to create the low-pressure region behind it leading to increased penetration, however, as a secondary effect, weak vortical structures induced by the pylon help enhance spreading and mixing.

**Ensemble-Averaged Structures**

Figures 4-4 through 4-7 show ensemble-averaged PLIF images for each injectant and injection pressure, with plane 1, plane 2, and plane 3 shown from left to right. For each image 300 single shot frames were used for averaging plane 1 and 2 and 600 frames were used for plane 3 since the latter showed weak intensities. The effects of injection pressure and molecular weight are described below.

**Effect of injection pressure**

Figure 4-4 shows the He injection with the pylon. At lower injection pressure, seen in Figure 4-4(A), for each plane the core of the plume is closer to the injection wall and the penetration is shorter than those in Figure 4-4(B), where the injection pressure was higher, and in planes 2 to 3 the spreading dominates with little increase of
penetration. At higher injection pressure, in plane 1 the core of the jet approaches the chamber centerline, at \( y/H = 0 \), in plane 2 the core of the triangularly shaped plume with wider spread is pushed toward the wall by the airflow expansion around the step and increases again after the reattachment point as shown in plane 3, a characteristic shape with top central part penetrating far into the core flow and even wider spread, almost reaching the side walls. The plume development for Ar injected behind the pylon, seen in Figure 4-5, shows a similar trend as He: higher injection pressure enhances penetration with the plume shape changed from triangular in plane 2 to the widely spread shape in plane 3.

Without the pylon, at higher injection pressure He injection, Figure 4-6 shows an elongated shape in plane 1 and becomes almost round further downstream. It is lifted from the injection wall with some increase of penetration to the step height and spreading, while at lower injection pressure a triangular shaped plume appears in plane 1 and at the end of recirculation region. Further downstream it remains close to the injection wall but it spreads more reaching the side walls. Figure 4-7 shows the plume images of Ar without pylon, with a similar development as He; at higher injection pressure the spreading is narrower but penetration is higher.

Penetration scales with the jet to freestream momentum flux ratio, \( J \), hence higher injection pressure increases the penetration regardless of the presence of pylon. Although previous studies have shown that the presence of pylons reduces spreading, here both penetration and spreading are increased and, furthermore, penetration is increased at higher injection pressure. The additional effect on spreading is due, likely,
to the presence of 3-D flow structure following expansion around the 2-D step as a result of the vortical motion induced by the presence of side walls.

**Effect of molecular weight**

When the pylon is present the jet plume axial development is similar for He and Ar with several notable differences. In plane 1 close to the step Ar penetrates less than He whereas in the far-field, at plane 3, the penetration is much higher at lower injection pressure as shown in Figures 4-4 and 4-5. Without the pylon there is no penetration difference between the two injectants as shown in Figures 4-6 and 4-7 but Ar has a wider spreading and a larger plume area than He in every corresponding case. Thus, it appears that the molecular weight has only a small effect on the plume penetration in agreement with the observations of Portz and Segal\(^9\)\(^1\) and Burger et al.\(^9\)\(^2\), although the heavier injectant can enhance spreading even without the aid of the pylon.

**Geometrical features of jet plume**

Figures 4-8 to 4-10 show certain salient features of the ensemble-averaged images including the plume area, penetration and lateral spreading. The 10% contour of the maximum intensity was taken as the jet plume boundary. The plume penetration \(y\) was determined from the peak location of this contour, and the lateral spread \(\Delta z\) was determined from the widest extend of it; both were normalized by the step height. To avoid any noise in the data, the pixels inside the 30% contour were counted as the plume area normalized by the injector area.

For every case the plume area gradually became larger except for He from plane 2 to plane 3 at 2.4 atm with pylon and at 5.1 atm without pylon as shown in Figure 4-8. At higher injection pressure the plume area increased regardless of the pylon’s presence but with the pylon the increase was larger. The increase is most significant at
higher injection pressure with pylon from plane 1 to plane 2. Figure 4-9 shows the plume penetration. For Ar the penetration was higher at higher injection pressure and the presence of pylon enhanced it. For He, except in the case with pylon, at higher injection pressure the penetration showed the same trend as Ar, while in other cases the penetration decreased from plane 2 to plane 3. Figure 4-10 compares the plume spreading of the first two planes due to the presence of the side walls at plane 3 that might cause some limit. For both injectants, with the pylon present the spreading was narrower than without the pylon.
Figure 4-1. Schlieren photograph of the air flow without injection or pylon\textsuperscript{70}. The black lines indicate the positions of the laser sheet for subsequent PLIF.
Figure 4-2. Normalized pressure distribution at two different stagnation injection pressures. There is a slight pressure increase of 0.2 atm behind the pylon and no difference downstream the step in the combustion chamber. This indicates that the presence of pylon causes no significant pressure loss. A) normalized pressure rise for He injected, B) normalized pressure rise for Ar injected.
Figure 4-3. Instantaneous end-view PLIF images in three measurement planes for four different injection cases: plane 1 (left), plane 2 (center), and plane 3 (right). Air flow direction is out of the paper plane, and the injection location is $z/H=0$ and $x/H=-10$, black solid line at $y/H=-1$ represents the step height. A) He-injection with pylon, B) He-injection without pylon, C) Ar-injection with pylon, D) Ar-injection without pylon.
Figure 4-3. Continued
Figure 4-3. Continued
Figure 4-4. Averaged end-view PLIF images for He-injection with-pylon cases at two different injection pressure: A) $P_{oj}=2.4$ atm and B) $P_{oj}=5.1$ atm. Images in planes 1, 2 and 3 are shown from left to right, air flow direction is out of the paper plane, and the injection location is $z/H=0$ and $x/H=-10$. The solid line at $y/H=-1$ represents where the step height is.
Figure 4-5. Averaged end-view PLIF images for Ar-injection with-pylon cases at two different injection pressure: A) \( P_0 = 2.4 \) atm, B) \( P_0 = 5.1 \) atm
Figure 4-6. Averaged end-view PLIF images for He-injection without-pylon cases at two different injection pressure: A) $P_0=2.4$ atm, B) $P_0=5.1$ atm
Figure 4-7. Averaged end-view PLIF images for Ar-injection without-pylon cases at two different injection pressure: A) $P_0 = 2.4$ atm, B) $P_0 = 5.1$ atm
Figure 4-8. Plume area comparison for each four cases along the streamwise direction for A) He and B) Ar. In order to obtain the area, the pixels within the contour of 30% value of the maximum intensity in each ensemble-averaged image were counted.
Figure 4-9. Plume penetration comparison for each four cases along the streamwise direction for A) He and B) Ar. The plume penetration was determined by the peak location of the 10% contour of the jet plume.
Figure 4-10. Plume spreading comparison for each four cases along the streamwise direction for A) He and B) Ar. The lateral spread $\Delta z/H$ was determined from the widest extend of the 10% contour of the jet plume.
CHAPTER 5
ISOLATOR LENGTH DERIVATION

With heat release in the combustion chamber, the pressure rises, shock system adjusts itself with the effect of propagating in an upstream direction. Once the shock train appears in the isolator, it can be pushed entirely out of it, which will cause the engine unstarts. However, isolator adds weight, internal drag and heat loads on the engine structure, therefore, its length must be limited to the minimum necessary required by operability constraints.

As a result of the formation of the shock train structure in the isolator, shown in Figure 1-5, the issue of immediate interest is the prediction of the minimal isolator length that can be allowed. In the figure, three regions are contained. In region 1 of the oblique shock train, the flow experiences a pressure gradient by increasing area contraction in supersonic region and by area expansion in subsonic region; and in region 3 the separation appears which balances the pressure gradient by shear stress on the outer boundary, which is region 2. For balance, the pressure gradient in the core flow must equal the shear stress, a relation between these two is established.

Anderson et al. employed a distortion-based methodology to solve the multidimensional fluid-dynamic phenomena by a one-dimensional context, a time-independent spatially one-dimensional integral formulation of the fluid-equation set in flux conservation form with arbitrary source terms is shown as:

\[
\frac{\partial}{\partial x} \begin{cases}
\rho u \eta_A \\
PA + \rho u^2 \eta_F \\
\rho u \eta_A H_0
\end{cases} = \begin{pmatrix}
\dot{s}_1 \\
\dot{s}_2 \\
\dot{s}_3
\end{pmatrix}
\]

where
- $P$ - static pressure,
- $\rho$ - density,
- $u$ - axial velocity,
- $A$ - geometric cross-sectional area,
- $H_0$ - bulk total enthalpy,
- $\dot{s}_1$ - mass source term,
- $\dot{s}_2$ - momentum source term,
- $\dot{s}_3$ - energy source term,
- $\eta_A$ - area distortion parameters (core and boundary layer) = $1 - \delta/A$ where $\delta$ equals the area defect of the flowfield,
- $\eta_F$ - momentum distortion parameters (core and boundary layer) = $1 - \delta/A - \theta/A$ where $\theta$ equals the momentum defect of the flowfield,
- $x$ - spatial coordinate.

Equation 5.1 quantifies the mass, stream thrust, and total enthalpy fluxes defined by the corresponding control surface.

The problem achieves closure assuming a linear dependence of the pressure gradient with the momentum and the wall friction coefficient.

$$\frac{dP}{dx} = 44.5C_{f0} \frac{\rho u^2}{D_H} \tag{5.2}$$

Where
- $C_{f0}$ - the coefficient of wall friction at the isolator entrance,
- $D_H$ - hydraulic diameter.
Isolator Length Derivation without Mass Source Addition

Commonly, in the one-dimensional control volume, the integrated values of mass, momentum, and energy fluxes are analyzed while concurrently employing an equation of state consistent with an equilibrium thermodynamic chemistry assumption when the source terms \( \dot{s}_1, \dot{s}_2, \dot{s}_3 \) are zeros, so Eq. 5.1 becomes:

\[
\dot{m} = \rho u \Lambda \eta_A
\]

(5.3)

\[
F = PA + \rho u^2 A
\]

(5.4)

\[
h_0 = h + \frac{u^2}{2}
\]

(5.5)

along with the Ideal gas equation: \( P = \rho RT \)

(5.6)

assuming that the mass and momentum defects are identical

\[
\eta_A = \eta_F
\]

(5.7)

Integrate Eq.5.2 from entrance \( x = 0 \) where the static pressure is \( P_1 \) to point \( x \) where the static pressure is \( P \)

\[
\int_0^x \frac{1}{\rho u^2} dP = \int_0^x \frac{44.5 C_{f0}}{D_H} dx
\]

\[
\frac{44.5 C_{f0}}{D_H} x = \int_0^x \frac{1}{\rho u^2} dP
\]

Assuming \( k = \rho u^2 \), the equation can be written in the form

\[
\frac{44.5 C_{f0}}{D_H} x = \int_0^x \frac{1}{k} dP
\]

(5.8)

From Eq.5.3 the following equation is obtained:

\[
\rho u = \frac{\dot{m}}{\Lambda \eta_A}
\]

(5.9)
Substitution of Eq. 5.7 and 5.9 into Eq. 5.4 results in the following equation:

\[ F = PA + \frac{m}{A \eta_A} u \eta_A \]

\[ = PA + m u \]

then

\[ u = \frac{F - PA}{m} \]  \hspace{1cm} (5.10)

For adiabatic flow:

\[ T_0 = T \left( 1 + \frac{\gamma - 1}{2} M^2 \right) \]

\[ M^2 = \frac{2}{\gamma - 1} \left( \frac{T_0}{T} - 1 \right) \]  \hspace{1cm} (5.11)

From ideal gas Eq. 5.6 and Mach number definition \( M = u / \sqrt{\gamma RT} \), \( \rho u^2 \) can be expressed as

\[ \rho u^2 = \gamma PM^2 \]  \hspace{1cm} (5.12)

Substitution of Eq. 5.6 and 5.11 into Eq. 5.12 the following equation is obtained:

\[ \rho u^2 = \frac{2 \gamma}{\gamma - 1} P \left( \frac{T_0 \rho R}{P} - 1 \right) \]

which is

\[ k = \frac{2 \gamma}{\gamma - 1} P \left( \frac{T_0 R}{P} \rho - 1 \right) \]

\[ = \frac{2 \gamma}{\gamma - 1} P \left( \frac{T_0 R}{P} \rho u^2 - 1 \right) \]

Substitution of Eq. 5.10 in the above equation, one can write

\[ k = \frac{2 \gamma}{\gamma - 1} P \left[ \frac{T_0 R}{P} k \left( \frac{m}{F - PA} \right)^2 - 1 \right] \]
\[ k = \frac{2\gamma}{\gamma-1} T_0 R k \left( \frac{\dot{m}}{F-PA} \right)^2 - \frac{2\gamma}{\gamma-1} P \]
\[ \frac{2\gamma}{\gamma-1} P = k \left[ \frac{2\gamma}{\gamma-1} T_0 R \left( \frac{\dot{m}}{F-PA} \right)^2 - 1 \right] \]
\[ \frac{1}{k} = \frac{1}{P} k T_0 R \left( \frac{\dot{m}}{F-PA} \right)^2 - \frac{2\gamma}{\gamma-1} \frac{1}{P} \]
\[ = m^2 T_0 R \left( \frac{1}{F-PA} \right)^2 \frac{1}{P} - \frac{2\gamma}{\gamma-1} \frac{1}{P} \]

Then Eq.5.8 becomes the following form,
\[ \left(\frac{44.5 C_{f0}}{D_H} \right) x = \dot{m}^2 RT_0 \int_{\bar{P}}^{P} \left( \frac{1}{F-PA} \right)^2 \frac{1}{P} dP - \frac{\gamma-1}{\gamma} \int_{\bar{P}}^{P} \frac{1}{P} dP \]

\[ = \dot{m}^2 RT_0 \int_{\bar{P}}^{P} \left( \frac{1}{F-PA} \right)^2 \frac{1}{P} dP - \frac{\gamma-1}{\gamma} \ln \frac{P}{P_1} \]

For the term \( \int_{\bar{P}}^{P} \left( \frac{1}{F-PA} \right)^2 \frac{1}{P} dP \), it can be derived as
\[ \int_{\bar{P}}^{P} \left( \frac{1}{F-PA} \right)^2 \frac{1}{P} dP = \int_{\bar{P}}^{P} \left( \frac{1}{PA} \right)^2 \frac{1}{F} \left( \frac{PA}{F} \right) d \left( \frac{PA}{F} \right) \]

Assuming \( t = \frac{PA}{F} \)

Equation 5.15 becomes
\[ = \frac{1}{F^2} \int \left[ \left( \frac{1}{1-t} \right)^2 + \frac{1}{t} + \frac{1}{1-t} \right] dt \]
\[ = \frac{1}{F^2} \left[ \frac{t}{1-t} + \ln t - \ln (1-t) \right]_{\bar{P}}^{P} \]
\[ = \frac{1}{F^2} \left[ \frac{t}{1-t} + \ln \frac{t}{1-t} \right]_{\bar{P}}^{P} \]
Substitution of $t = \frac{PA}{F}$ back, the following form can be obtained,

$$
= \frac{1}{F^2} \left[ \frac{1}{1 - \frac{PA}{F}} + \ln \frac{PA}{F} \right]_{l_1}
$$

$$
= \frac{1}{F^2} \left[ \frac{F}{A} + \ln \frac{P}{F} \right]_{l_2}
$$

$$
= \frac{1}{F^2} \left[ \frac{F}{A} - P - \frac{F}{A} (\frac{F}{A} - P) \frac{P}{F} - \frac{F}{A} (\frac{F}{A} - P) \frac{1}{P_i} \right] + \ln \frac{F}{P_i} \frac{P}{F}
$$

$$
= \frac{1}{F^2} \left[ \frac{F}{A} (P - P_i) \frac{1}{P_i} \right] + \ln \frac{F}{P_i} \frac{P}{F}
$$

$$
= \frac{1}{F^2} \left[ \frac{F}{P_i} - P \frac{1}{P_i} \right] + \ln \frac{F}{P_i} \frac{P}{F}
$$

$$
= \frac{1}{F^2 P_i} \left[ \frac{F}{P_i} - P \frac{1}{P_i} \right] + \ln \frac{F}{P_i} \frac{P}{F}
$$

$$
\text{Assuming} \ P_r = \frac{P}{P_i}, \ C_b = \frac{F}{P_i}
$$

Equation 5.16 results in the form:

$$
= \frac{1}{FAP} \left[ \frac{P - P_r}{(C_b - P_r)(C_b - 1)} + \ln \frac{P}{C_b} \frac{(C_b - 1)}{P_r} \right]
$$

$$
= \frac{1}{FAP} \left[ \frac{P - P_r}{(C_b - P_r)(C_b - 1)} - \ln \frac{C_b - P_r}{P_r} \frac{(C_b - 1)}{P_r} \right]
$$

The result of Eq.5.14 is
Assuming

\[ C_a = \frac{\dot{m}^2}{P_1 A} \sqrt{\frac{R T_0}{\gamma}} \quad \xi = \frac{44.5 C_{f_0}}{D_H} x \]

The final solution of Eq. 5.2 is given by

\[ \dot{\xi} = \gamma \frac{C_a^2}{C_b} \left[ \frac{P_r - 1}{P_r (C_b - P_r)} - \frac{1}{C_b} \ln \left( \frac{C_b - P_r}{P_r (C_b - 1)} \right) \right] - \frac{\gamma - 1}{2\gamma} \ln P_r \]  

(5.18)

Where

\[ P_r = \frac{P}{P_1}, \quad C_b = \frac{F}{P_1 A}, \quad C_a = \frac{\dot{m}^2}{P_1 A} \sqrt{\frac{R T_0}{\gamma}}, \quad \xi = \frac{44.5 C_{f_0}}{D_H} x \]

**Isolator Length Derivation with Mass Source Addition**

Equation 5.18 was obtained for the isolator flow without any source addition, but sometimes to increase the fuel residence time and improve the mixing efficiency in the combustion chamber; part of the fuel may be injected upstream, in the isolator. In this case, the mass source addition \( \dot{s}_1 \neq 0 \), and the momentum and energy source terms \( \dot{s}_2 \neq 0, \dot{s}_3 \neq 0 \), so the integration of Eq. 5.1 can be obtained as

\[ \rho u A \eta_A = \dot{m} + \dot{s}_1 x \]  

(5.19)

\[ P A + \rho u^2 A \eta_F = F + \dot{s}_2 x \]  

(5.20)

\[ h + u^2/2 = h_0 + \dot{s}_3 x \]  

(5.21)
Equation 5.19 can be written as

\[ \rho u = \frac{\dot{m} + \dot{s}_x}{A \eta_A} \]  

(5.22)

Substitution of Eq. 5.7 and 5.22 into Eq. 5.20, it becomes

\[ PA + \frac{\dot{m}}{A \eta_A} u A \eta_A = F + \dot{s}_x \]

then

\[ u = \frac{F + \dot{s}_x - PA}{\dot{m} + \dot{s}_x} \]  

(5.23)

Expansion of Eq. 5.21, the following results can be obtained,

\[ c_p T + \frac{u^2}{2} = c_p T_0 + \dot{s}_x \]

\[ \frac{\gamma}{\gamma - 1} RT + \frac{u^2}{2} = \frac{\gamma}{\gamma - 1} RT_0 + \dot{s}_x \]

\[ T_0 = T + \frac{\gamma - 1}{\gamma R} \frac{u^2}{2} - \frac{\gamma - 1}{\gamma R} \dot{s}_x \]

\[ T_0 = T \left( 1 + \frac{\gamma - 1}{2} M^2 \right) - \frac{\gamma - 1}{\gamma R} \dot{s}_x \]

\[ M^2 = \frac{2}{\gamma - 1} \left( \frac{T_0 + \frac{\gamma - 1}{\gamma R} \dot{s}_x}{T} - 1 \right) \]  

(5.24)

Substitution of Eq. 5.6 and 5.24 into Eq. 5.12 results of the following equation:
\[ \rho u^2 = \frac{2\gamma}{\gamma - 1} P \left( \frac{T_0 + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{T} - 1 \right) \]

\[ = \frac{2\gamma}{\gamma - 1} P \left( \frac{T_0 + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \right) \left( \frac{\gamma}{P} \right) (\rho - 1) \]

\[ = \frac{2\gamma}{\gamma - 1} P \left( \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \right) \left( \rho u^2 \frac{1}{u^2} - 1 \right) \] 

(5.25)

Assuming \( k = \rho u^2 \) and substitute Eq. 5.23 into Eq. 5.25

\[ k = \frac{2\gamma}{\gamma - 1} P \left( \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \right) \left( \frac{(\dot{m} + \dot{s}_1 x)^2}{(F + \dot{s}_2 x - PA)^2} - 1 \right) \]

\[ \frac{2\gamma}{\gamma - 1} P = k \left[ \frac{2\gamma}{\gamma - 1} \left( \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \right) \left( \frac{(\dot{m} + \dot{s}_1 x)^2}{(F + \dot{s}_2 x - PA)^2} - 1 \right) \right] \]

\[ \frac{1}{k} = \frac{1}{P} \left( \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \right) \left( \frac{1}{(F + \dot{s}_2 x - PA)^2} \right) \left( \frac{(\dot{m} + \dot{s}_1 x)^2}{P} - \frac{2\gamma}{\gamma - 1} \frac{1}{P} \right) \] 

(5.26)

Then Eq. 5.2 becomes

\[ \left[ \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \left( \frac{(\dot{m} + \dot{s}_1 x)^2}{P} \right) \left( \frac{\gamma - 1}{2\gamma} \frac{1}{P} \right) \right] dp = \frac{44.5 C_{f_0}}{D_H} dx \]

\[ \frac{dx}{dP} = \frac{D_H}{44.5 C_{f_0}} \left( \frac{T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_3 x}{P} \left( \frac{(\dot{m} + \dot{s}_1 x)^2}{P} \right) \right) \left( \frac{\gamma - 1}{2\gamma} \frac{1}{P} \right) - \frac{D_H \left( \frac{\gamma - 1}{89} \frac{1}{P} \right)}{89 C_{f_0}} \] 

(5.27)

Assuming \( a = \frac{D_H}{44.5 C_{f_0}} \), \( y = x \), \( x = P \)
\[
\frac{dy}{dx} = a \left( T_0 R + \frac{\gamma - 1}{\gamma} \dot{s}_1 y \right) \frac{(\dot{m} + \dot{s}_1 y)^2}{(F - Ax + \dot{s}_2 y)^2} - \frac{a(\gamma - 1)}{2\gamma} \frac{1}{x}
\]

\[
\frac{dy}{dx} = a \gamma \frac{\gamma - 1}{\gamma} \dot{s}_1^2 \dot{s}_3 y^3 + \left( \dot{s}_1 T_0 R + \frac{2(\gamma - 1)}{\gamma} \dot{m} \dot{s}_1 \dot{s}_3 \right) y^2 + \left( 2\dot{m} \dot{s}_1 T_0 R + \frac{(\gamma - 1)}{\gamma} \dot{m}^2 \dot{s}_3 \right) y + \dot{m}^2 T_0 R \frac{1}{(F - Ax + \dot{s}_2 y)^2} - \frac{a(\gamma - 1)}{2\gamma} \frac{1}{x}
\]

Assuming

\[ f_3 = \frac{\gamma - 1}{\gamma} \dot{s}_1^2 \dot{s}_3 \]

\[ f_2 = \dot{s}_1 T_0 R + \frac{2(\gamma - 1)}{\gamma} \dot{m} \dot{s}_1 \dot{s}_3 \]

\[ f_1 = 2\dot{m} \dot{s}_1 T_0 R + \frac{(\gamma - 1)}{\gamma} \dot{m}^2 \dot{s}_3 \]

\[ f_0 = \dot{m}^2 T_0 R \]

Equation 5.28 becomes

\[
\frac{dy}{dx} = a f_3 y^3 + f_2 y^2 + f_1 y + f_0 \frac{1}{x} - \frac{a(\gamma - 1)}{2\gamma} \frac{1}{x}
\]

Equation 5.28 is a first order, non-linear differential equation, but it does not have a solution in closed form. Furthermore, with mass injection in the isolator, the shock pattern is changed, so the isolator flow cannot be treated as a one-dimensional flow any more, and it should be considered three-dimensional. The solution can be found only by numerical derivation. One possible open-literature package is ANSYS Fluent which is employed in Chapter 6.
CHAPTER 6
FLUENT SIMULATION: SHOCK WAVE/BOUNDARY-LAYER INTERACTION

ANSYS Fluent software, one of the most comprehensive software tools available to the computational fluid dynamics (CFD) research area, uses the finite-volume method to solve the governing equations. It contains a broad range of physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants\textsuperscript{93,94}.

Fluent includes two numerical methods: pressure-based solver and density-based solver. For low-speed incompressible flows, the pressure-based solver was used, while for high-speed compressible flows, the density-based solver was employed. However, recently both methods have been extended and reformulated to solve and operate for a wide range of flow conditions beyond their traditional intent\textsuperscript{95}. In both methods, Fluent will solve the Navier-Stoke’s governing integral equations for the conservation of mass, momentum, energy, and other scalar elements such as turbulence, dissipation, chemical species, etc. But for supersonic cases, density-based method is preferred; the overview of the method is shown in Figure 6-1\textsuperscript{95}.

Fluent employs a control-volume-based technique to convert a general scalar transport equation to a numerically solved algebraic equation. This technique integrates the control volume’s transport equation, and generates a discrete equation which can be expressed by the conservation law on a control-volume basis.

There are many models in Fluent, Spalart-Allmaras Model, Standard, RNG, and Realizable $k$-$\varepsilon$ models, standard and SST $k$-$\omega$ models, Reynolds stress model (RSM),
Large eddy simulation (LES) model, etc. User chooses the proper model according to the simulation situation. Here, after comparison with several models, $k-\varepsilon$ RNG model was used.

From the schlieren photography experiments, we got three Mach number’s flow structure development in isolator and test section at certain percentage of exit area reduction. When the pressure in the test section rises, the shock system adjusts itself with the effect of propagating in an upstream direction, when the shock train appears in the isolator, at certain blockage, the shock train reaches its steady state, followed by separation of boundary layer.

The 2-D model of the experimental facility was built by Gambit and computed in Fluent at selected blockage 60% at Mach 1.9, the results were compared with the experimental results.

The 2-D model geometry is shown in Figure 6-2, the grid distribution is shown in Figure 6-3, to capture the shock, 100 grids in 1 inch was applied, and near the wall, the mesh for the boundary layer is finer.

For the computation, steady, compressible, $k-\varepsilon$ RNG (Re-Normalisation Group) model has been applied with two different near-wall treatment, (a) enhanced wall treatment and (b) standard wall function. At the inlet, static pressure is $P_s=9.819$ psi, $T_0=300$ K, $P_0$ is interpolated from experimental measurements at the isolator entrance (Figure 3-4), the profile after interpolation is shown in Figure 6-4, hydraulic diameter = 1 in. At the outlet, static pressure is atmosphere pressure 14.7 psi, $T_0=300$ K, hydraulic diameter = 0.8 in.
With the enhanced wall treatment used, the computing procedure for the Mach number contour is shown in Figure 6-5. The oblique shock appears in isolator, but it turns rapidly into normal shock and eventually moves out of isolator before the computation converges, which is different from the experimental results.

Changing the model to \( k-\varepsilon \) RNG with standard wall function and wall roughness height 8.04E-5 inches because the facility is built of brass\(^{96} \), the computation procedure of Mach number contours is shown in Figure 6-6, we can see the bifurcated shock appears in the isolator followed by separation at the foot of leading oblique shock. In the third image, behind the oblique shock, there are several regions with \( M>1 \), it seems normal shocks, which are similar to the results in experiments, but the bifurcated shock eventually turns to normal shock and runs out of isolator before the computation becomes steady.

The computation results are different from the experimental ones may due to two reasons, first is the actual wall friction force, results in Figure 6-5 are obtained with enhanced wall treatment which does not consider wall roughness, and in Figure 6-6 wall roughness is considered, bifurcated shock stays in the isolator for a while before it turns to normal shock, which means the wall friction force has a very important effect on the shock pattern and its stabilization, and this can also be seen from the isolator wall pressure rise prediction in Chapter 3. The second reason may be the model used, here a 2-D model was applied. In fluent, 2-D model is actually computed in a 3-D model with third direction length of 1 m, while the actual model has only 1 inch in the third direction, and from the isolator entrance total pressure profile in Figure 3-4 the third direction has much thicker boundary layer, which may have some effect on the shock stabilization.
The discrepancy can be due to uncertainty of the proper wall friction force. A 3-D model should, then, be employed.
Figure 6-1. Overview of the density-based solution method

Figure 6-2. 2-D geometry model used in Fluent computation
Figure 6-3. Mesh distribution in the model

Figure 6-4. Inlet total pressure profile after interpolation
Figure 6-5. Computing procedure of Mach number contour for $M=1.9$ at 60% blockage, using $k-\varepsilon$ RNG model with enhanced wall treatment.
Figure 6-6. Computing procedure of Mach number contour for M=1.9 at 60% blockage, using k-ε RNG model with standard wall function, wall roughness height=8.04E-05 in.
CHAPTER 7
CONCLUSIONS AND FUTURE WORK

The focus of this work is to evaluate isolator/combustion chamber interactions, which include the effects of pressure rise in the combustion chamber on the isolator flow, and the mixing effects - the effects of mixing schemes on the combustion chamber mixing in the non-reacting flow. Schlieren photography method and acetone planar laser induced fluorescence method were used to obtain the images in the experiments for analysis and evaluation respectively. Fluent computation was applied for the shock wave/Boundary layer interaction. The conclusions are summarized below.

Isolator/Combustion Chamber Interactions

Effects of isolator/combustion chamber interactions were studied experimentally at Mach numbers of 1.6, 1.9 and 2.5 respectively.

- At Mach 1.6, the shock train system consists of a series of symmetric, nearly normal shocks. The first shock is bifurcated with incipient separation and is followed by several weaker secondary unbifurcated nearly normal shocks which did not separate the boundary layer. When clear separation of the boundary layer along the isolator walls is noticed, the shocks can be pushed entirely upstream of the isolator and the test section is almost entirely operating in a subsonic mode. Any further heat release should be avoided to result in upstream effects in the inlet with unstart.

- With Mach number increase to 1.9, the shock train interaction with the boundary layer is stronger and the point of bifurcation of the leading shock moves away from the wall and reaches the centerline of the duct. The normal part of the leading shock disappeared, forming an oblique shock train.

- In contrast, the Mach 2.5 shock train was an asymmetric, oblique shock system, with less apparent bifurcated shocks but increased separation region.

- Taking into account the correct modified deficit of the entrance stream thrust and mass flowrate, the pressure rise was predicted within 10%. Other factors, in particular the proper selection of wall friction coefficient, play a significant role.
Non-Reacting Mixing

Upstream pylon-aided injection into a Mach 1.6 air stream has been studied using PLIF with data recorded at three planes in the combustion chamber behind a two-dimensional step; the planes were located in the recirculation region and beyond, at 0.5H, 2H, and 10H respectively. Injection pressures, 2.4 atm and 5.1 atm, and injectants molecular weight, light molecular helium and heavy molecular argon, were examined with emphasis on penetration, spreading and shape of the jet plumes. The results showed the following:

- The presence of thin pylon causes essentially no pressure loss.
- With the pylon all the jet is lifted from the injection wall with both penetration and spreading increasing. Penetration is increased more at higher injection pressure while spreading dominates at lower injection pressure.
- Without the pylon the injectant penetration relies only on the injection pressure but the injectants remain close to the wall with considerably increased spreading at the lower injection pressure.
- The injectant molecular weight has little effect on the jet penetration but the heavier injectant shows increased spreading when the pylon is absent.
- In the near-field the presence of the pylon leads to increased penetration and reduced spreading; however, in the far-field spreading is improved by other factors, notably by the large vortical structures induced by the presence of side walls.

Isolator Length Derivation

The relation between isolator length and pressure rise was derived by assuming adiabatic constant-area isolator flow one-dimensional.

- Without mass addition in the isolator, all the source terms are zeros. A closed form solution was obtained which can be applicable only in the region with flow separation.
- With the mass addition in the isolator, all the source terms are non-zeros. The non-linear differential equation does not have a solution in closed form, which
may be due to the disturbance/change of the shock pattern with the fuel injection, the flow cannot be assumed as on-dimensional.

**Shock Wave/Boundary-Layer Interaction Simulation**

The model of 60% blockage at M=1.9 was simulated using k-ε RNG model with two different near wall treatment.

- Model with enhanced wall treatment does not consider wall roughness, bifurcated shock appears in isolator but turns to normal shock rapidly, and the shock wave runs entirely out of isolator before the computation converges.

- In the model with standard wall function, wall roughness is considered. Bifurcated shock appears with separation followed, which lasts for a certain distance when the computation proceeds, but eventually cannot stabilized in the isolator.

- Proper actual wall friction force has a very important effect on the shock stabilization.

**Future Work**

The PLIF tests were operated at the front windows which gave the information of fuel mixing and penetration in the direction normal to the streamwise, so the PLIF images and information along the streamwise direction need to be obtained.

The isolator length derivation needs further investigation when the fuel is injected; since the one-dimensional flow assumption cannot be used to simulate the actual flow, 3-D analysis is required.

This research has focused on the non-reacting cold air flow, which can provide some useful information about the effect of various parameters to the actual combustion, like operating conditions, fuel injection pressure and locations, etc. But these data are limited since combustion of fuel-air mixture does not take place. So the combustion test (OH-PLIF) needs to be performed, the flameholding stability, stoichiometric ratio of the burning and other parameters needs to be studied.
The effect of proper wall friction force on the shock wave/ boundary-layer interaction simulation needs further analysis in 3-D models.
APPENDIX A
DATA ACQUISITION AND CONTROL HARDWARE AND NATIONAL INSTRUMENTS HARDWARE

The data acquisition system and hardware used in the scramjet facility were listed clearly by Owens. Figure A-1 shows a schematic of the data acquisition and control (DAQC) hardware. The DAQC hardware consists of the instrumentation installed on the facility and the computer hardware used to acquire and/or output instrumentation signals. Details on the specific devices are given in Table A-1.

Figure A-1. Data acquisition and control hardware

The hardware for acquiring / output of the instrumentation signals consists of:

1. A Pentium based computer.

2. A National Instruments AT-MIO-16E-2 data acquisition board installed in the computer, featuring 8 differential input channels and a 500 kHz maximum scan rate.


5. A National Instruments PCII/IIA GPIB interface board installed in the computer.

Instrumentation signals are acquired from five types of devices:

1. Four Omega PX303 absolute pressure transducers generating 0.5 to 5.5 volt signals that are proportional to the pressure sensed and are read via the AI-MIO-16E-2 data acquisition board.

2. Twelve Omega k-Type thermocouples generating voltage signals that are a polynomial function of junction temperature and are read by the AI-MIO-16E-2 / SCXI combination.

3. One Sponsler MF30 turbine flowmeter generating a 0 – 2200 Hz signal that is proportional to the volumetric flowrate and is read by the AI-MIO-16E-2 data acquisition board.

4. A Pressure Systems PSI-9010 pressure scanner featuring a bank of 16 gage-pressure transducers that is read via the RS-232 serial port.

5. A Mensor DPG II 200-psia absolute pressure transducer that is read via the PCII/IIA GPIB interface board.

   Additionally, setpoint voltages are output to two Tescom ER – 3000 electronic PID controllers.

The air stagnation pressure is read by the Mensor DPG II pressure transducer.

The air static pressure at the nozzle exit and the test section wall pressures are measured by the Pressure Systems PSI 9010 pressure scanner. The H₂ and O₂ stagnation pressures in the air-heater and the fuel pressures are each measured by an
Omega PX303 200-psia pressure transducer. Omega k-type thermocouples are used to measure the 4 temperatures used to determine the air stagnation temperature, the 2 air-heater wall temperatures, and the 5 test section wall temperatures.

Table A-1. Instrumentation specifications

<table>
<thead>
<tr>
<th>Manufacturer and Model</th>
<th>Measure and Sensed</th>
<th>Controlled</th>
<th>Range</th>
<th>Input</th>
<th>Output</th>
<th>Feedback</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Systems PSI 9010 Pressure Scanner</td>
<td>$P_{air}$, $P_{w(\frac{X}{H})}$</td>
<td>0 - 30 psig</td>
<td>PC Serial Port</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Omega Engineering PX303 Pressure Transducer</td>
<td>$P_{t,H2}$, $P_{t,O2}$, $P_{t,H2-pilot}$, $P_{t,K}$</td>
<td>0 - 200 psia</td>
<td>0 - 5 V analog</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Omega Engineering K-Type Thermocouple</td>
<td>$T_{t,air}$, $T_{t,K}$, $T_{w(\frac{X}{H})}$,</td>
<td>300 – 2000 K</td>
<td>0 – 50 nV analog</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mensor DPG II Pressure Transducer</td>
<td>$P_{t,air}$</td>
<td>0 – 200 psia</td>
<td>PC GPIB Board</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tescom ER-300 Electronic Pressure Controller</td>
<td>Tescom 4400</td>
<td>0.5 – 5.5 V analog</td>
<td>0 – 100 psig $N_2$</td>
<td>0.5 – 5.5 V analog</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tescom 4400 Seres Dome-Loaded Pressure Regulator</td>
<td>$P_{t,H2}$, $P_{t,O2}$, $P_{t,H2-pilot}$, $P_{t,K}$</td>
<td>0 - 60 psig $N_2$</td>
<td>0 – 600 psig $H_2$, $N_2$, or $O_2$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tescom 1300 Series Manual Pressure Regulator</td>
<td>Tescom 4400</td>
<td>Manual Operation</td>
<td>0 - 100 psig $N_2$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ametek PMT Model 40 Pneumatic Controller</td>
<td>Fisher Model ED</td>
<td>Dial Selectable</td>
<td>0 – 30 psig $N_2$</td>
<td>0 – 150 psig</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fisher Controls Type ED Dome-loaded Butterfly Valve</td>
<td>$P_{t,air}$</td>
<td>0 – 300 psig</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>National Instruments AT-MIO-16-E2 DAQ Board</td>
<td></td>
<td>±10 V analog (8)</td>
<td>±10 V analog (2)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manufacturer and Model</td>
<td>Measure and Sensed</td>
<td>Controlled</td>
<td>Range</td>
<td>Input</td>
<td>Output</td>
<td>Feedback</td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------</td>
<td>------------</td>
<td>-------</td>
<td>-------</td>
<td>--------</td>
<td>----------</td>
</tr>
<tr>
<td>National Instruments PC II/Ila GPIB Controller Board</td>
<td>Mensor</td>
<td>DPG II</td>
<td>Pressure</td>
<td>Transducer</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table A-1. Continued.
APPENDIX B
WALL FRICTION COEFFICIENT CALCULATION FORMULA

\[ C_{f0} = \frac{1.325}{\ln \left( \frac{l}{3.7D_H} + \frac{5.74}{Re^{0.69}} \right)^2} \]

Where

- \( l \) - length of the duct
- \( D_H \) - hydraulic diameter of the duct,
- \( Re = 3.02 \times 10^8 \) at Mach 1.9,

When \( l = 5 \times 10^{-6} \) ft which the isolator length used in this research, the wall friction coefficient is \( C_{f0} = 0.0109 \).

When \( l = 0 \) ft which is the ideal case, the wall friction coefficient is \( C_{f0} = 0.005 \).\(^*\)

\(^*\) Adapted from Moody Friction Factor Calculator, [http://www.lmnoeng.com/moody.htm](http://www.lmnoeng.com/moody.htm)
APPENDIX C
SCRAMJET ACTUATOR

The actuator mechanism used in the scramjet facility to block the test section exit was manufactured by Precise Motion & Control Inc. Below is the company report about the working principles, codes and the LabView program.

**Working Principles and Sequence of Codes for Actuator**

The motion for the linear actuator follows a trapezoidal profile. Thus the profile has three portions, a positive slope representing Acceleration (AC), a flat portion or the plateau representing constant velocity (VE) and the third a negative slope representing deceleration (DE).†

![Trapezoidal motion profile of the linear actuator](image)

Figure C-1. Trapezoidal motion profile of the linear actuator

† Adapted from Reports from Precise Motion & Control Inc, [www.precisemotion.com](http://www.precisemotion.com)
A sequence of strings was generated using LabView and sent to the STAC6-Q controller/driver through the serial port of the computer. These strings are program sequences understood by the driver to perform guided motion of the stepper motor. These guided motions include the direction of motion of the motor shaft (clockwise and counter clockwise) as well as the distance that the linear actuator would advance or recede depending on command sequence passed to the driver.

**Controlling the motion of the actuator arm**

The actuator used in the lab has a minimum lead/advance of 0.2 inches corresponding to a micro step resolution of 25,000 step/resolution of the stepper rotor. Hence, to produce a 12 inch advance/recession of the actuator arm, we need $(12/0.2*25000)=1,500,000$ steps. This motion is brought out at prescribed acceleration (positive slop OA of the Figure C-1), constant maximum velocity (AB portion of Figure C-1) and deceleration (Slop BC in a trapezoidal profile as mentioned in Figure C-1). Hence the sequence of commands in LabView ("Basic Serial and Drive.VI" file) includes mentioning the Acceleration (AC), Maximum Velocity attained (VE), Deceleration (DE)
and the Distance/Steps (DI) needed to be moved. A typical sequence of LabView commands is shown below.

```
AC10\r
DE10\r
VE6\r
DI1500000\r
FL
```

Any parameter following the DI command recognizes the sign before it. A (+) sign would mean travel in one direction and a (-) sign would mean travel in the reverse direction. Hence, DI command is always followed by a (+/-) sign. The set up in this case is presently accepting DI-950000 to stretch the arm (hence close the area of the test section) and DI950000 to bring the actuator arm to the reference origin (hence bring the test area to no close or full open position). FL represents “Feed to Length” to carry out the mentioned steps. And \r corresponds to the ASCII equivalent of a “carriage return”.

A typical sequence of motion that closes the test section and then opens it back to original position is shown below

```
AC10\r
DE10\r
VE6\r
DI950000\r
FL
AC10\r
DE10\r
```
VE6\r
DI-950000\r
FL

Notice carefully the (-) sign following DI in the second last line that brings about the reverse motion. These values are real time and no fabrication. In other words, the setup is now operating exactly as per the commands shown above. Notice carefully that DI is (+/-) 950,000 and not (+/-) 1,500,000. A DI 1,500,000 was used for the first time only to provide for some clearance to the actuator arm.

The motion of the arm can be changed during any part of the trapezoidal profile, e.g., the velocity could be changed (to change the arm go faster or slower) using command sequence VC (corresponding to Velocity Change) and then using FC corresponding to the Feed needed to be produced with the changed velocity. A typical example follows,

DI1500000\r
VE6.0\r
DI1500000\r
VC1.0\r
FC\r

In the above example the first 12 inch step travel is brought about at a velocity of 6 revolutions/sec and the next 12 inches are brought about at a reduced/changed velocity of 1 revolution/sec. Here no acceleration or deceleration was chosen.

Controlling the Thrust Produced by the Actuator Arm

The thrust produced by the actuator arm comes factory specified. It depends on the winding current setting of the stepper motor, which in this case is set at drawing 3.5
Amp of current. However, this current setting and hence the thrust output can be changed by including an additional command line of “CC2.4”. This command means change current (CC) from the defraud setting of 3.5 Amps to 2.4 amps. The stepper motor responds linearly (factory specified calibration) to this current change setting i.e. the thrust reduces by a factor of (2.4/3.5). However, for this set-up, the arm is operating at the factory default thrust of 800N.
LabView Program for Actuator Controller

ScramjetActuatorControl.vi

Front Panel

Switch "Use calib" off and set "Counts set" when inputing direct counts to the actuator, e.g. during calibration.

 baud rate 9600
 VISA resource name COM1
 data bits 8 parity None
 stop bits 1.0 flow control None
 initialization string SP0\n
 counts from cal
 1278
 set counts 1278
 In Range? OK

 desired opening (inch)
 2

 GO

 reset string to COM
 D10\n FP\n
This program controls the actuator used for downstream blockage of the scrmjet setup.

Block Diagram

- Initialization string
- Flow control
- Stop bits
- VISA resource name
- baud rate
- data bits
- parity
This program controls the actuator used for downstream blockage of the scramjet setup.

Send move command to serial port.
This program controls the actuator used for downstream blockage of the scarpet setup.

Wait for user to enter steps to move and click a button.

**Flow control**

**Stop blockage**

**VISA resource name**

**Baud rate**

**Date/time**

**parity**

Use call

AC-ID

set counter

PER

desired opening (inch)

stop

TO

counts from cell
This program controls the actuator used for downstream blockage of the screenjet setup.
Read and write.vi

Front Panel

Select the serial resource and the operations (Read, Write, or both) to be performed. If both are selected, the VI will write the data first, read data and then close the VISA session that is opened to the port. This VI will wait until the specified number of bytes is received at the port. Only the number of bytes specified will be read.

For additional information select File >> VI Properties >> Documentation
Block Diagram

Enable Termination Char

timeout (milliseconds)

VISA resource name

baud rate

data bits

parity

stop bits

flow control

Configure Serial port (baud rate, data bits, parity, stop bits and flow control)

Write bytes to port

Read bytes at port

Delay before read (ms)

Read the number of bytes specified

Close session to port

Note: Closing the serial port allows it to be used by other applications instead.
Actuator calibration.vi

Front Panel

![Opening width (inch) 1.7]

![Actuator position (counts) 316626]

Block Diagram

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

Actuator position (counts) 316626

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

![Actuatorcal070917b.xls]

Opening width (inch) 1.7

![Actuatorcal070917b.xls]

Opening width (inch) 1.7
LIST OF REFERENCES


7 Ramjet Propulsion. NASA Glenn Research Center.


14 “Researchers put spark into scramjets”. Australian Broadcasting Corporation, ABC News. 16 November 2010.


94 http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+Fluent

95 ANSYS Fluent Theory Guide


99 Reports from Precise Motion & Control Inc, www.precisemotion.com
BIOGRAPHICAL SKETCH

Qiuya Tu was born in China, grew up in Nanyang City, Henan Province. In 2000, she enrolled at Beijing Institute of Technology and received the Bachelor of Science degree in Department of Flight Vehicle specialized in Propulsion in June 2004 with the 1st place in the department. In 2004 she joined the graduate school of Beijing Institute of Technology with the entrance exam waived and obtained her Master degree in Aircraft Design area in July 2006. In 2007 she enrolled in the Ph.D. program in Mechanical Engineering at the University of Florida and received her degree in the fall of 2013.