NUMERICAL SIMULATION OF CANTILEVERED RAMP INJECTOR FLOW FIELDS FOR HYPERVELOCITY FUEL/AIR MIXING ENHANCEMENT

by

Jurgen Schumacher

A thesis submitted in conformity with the requirements for the degree of Doctor of Philosophy
Graduate Department of Aerospace Science and Engineering
University of Toronto

Copyright © 2000 by Jurgen Schumacher
The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author’s permission.
NUMERICAL SIMULATION OF CANTILEVERED RAMP INJECTOR FLOW FIELDS FOR HYPERSONIC FUEL/AIR MIXING ENHANCEMENT

Jurgen Schumacher
Doctor of Philosophy
Graduate Department of Aerospace Science and Engineering
University of Toronto
2000

Abstract

Increasing demand for affordable access to space and high speed terrestrial transport has spawned research interest into various air-breathing hypersonic propulsion systems. Propulsion concepts such as the supersonic combustion ramjet (scramjet) and the shock-induced combustion ramjet (shcramjet) utilize oxygen freely available in the atmosphere and thereby substantially reduce the weight penalty of on-board oxidizer tankage used in rocket based systems. Of key importance to the ultimate success of an air-breathing concept is the ability to efficiently mix the fuel with atmospheric air. In the case of a hypersonic air-breather the challenge is accentuated due to the requirement of supersonic combustion. Flow velocities through the combustor on the order of thousands of meters per second provide the fuel and air with only a brief time to adequately combine. Contemporary mixing augmentation methods to address this issue have focused on fuel injection devices which promote axial vortices to enhance the mixing process. Much research effort has been expended on investigation of ramp injectors for this purpose. The present study introduces a new ramp injector design, based on the conventional ramp injector, dubbed the cantilevered ramp injector.

A two-pronged numerical approach was employed to investigate the mixing performance and characteristics of the cantilevered injector consisting of, 1) comparison with conventional...
designs and 2), a parametric study of various cantilevered injector geometries. A laminar, three-dimensional, multispecies flowsolver was developed in generalized coordinates to solve the Navier-Stokes equations for the flow fields of injected H\textsubscript{2} into high-enthalpy air. The scheme consists of an upwind TVD scheme for discretization of the convective fluxes coupled with a semi-implicit LU-SGS scheme for temporal discretization.

Through analysis of the numerical solutions, it has been shown that the cantilevered ramp injector is a viable fuel injection system facilitating enhanced mixing of fuel and air. Comparison with conventional designs have revealed a competitive and, in most cases, superior design in the context of mixing performance. A strong counter-rotating vortex pair generated under the cantilevered injector was shown to be the distinguishing characteristic of this design and largely accounted for improved mixing performance. Results also elucidated the importance of a coupled design approach between the fuel injector and propulsive duct to optimize mixing performance.
To my parents.
I would like to extend sincere thanks, first and foremost, to my thesis supervisor, Prof. J. P. Sislian, for his inspiring love of science and research, astute guidance and especially his friendship. His ability to rekindle intellectual excitement in research has been a continual source of strength.

The author would also like to convey his appreciation to all the faculty, staff and students at UTIAS for their help and support. Special thanks to my colleagues: Rudy, Samir, Giovanni, Bernard, Peter, Kevin, and many more for their friendship, selflessness and fruitful discussions during the course of this research. The mentoring efforts of Dr. Dudebout have been invaluable since the start of my graduate work.

Thanks to all those at the Shock Wave Research Center, Tohoku University: Prof. K. Takayama, Tetsuya Kodama and Mingyu Sun to name but a few. They have significantly enriched and broadened my educational experience through their contributions to this work, sharing of diverse prospective and knowledge, and their friendship.

The author would also like to gratefully thank his family for their unwavering support; they never failed to believe in me or to make me believe in myself.

Financial support from the National Science and Engineering Research Council, the Ontario Graduate Scholarship program and the University of Toronto is acknowledged.
Abstract

Acknowledgements

Nomenclature

1 Introduction

1.1 Background and Motivation ........................................... 1
1.2 Mixing Enhancement Strategies ........................................ 3
  1.2.1 Non-Intrusive Injectors: Orifices ............................... 3
  1.2.2 Strut Injectors ..................................................... 3
  1.2.3 Ramp Injectors ..................................................... 3
1.3 Axial Vortex Generation Mechanisms ................................. 4
1.4 The Cantilevered Ramp Fuel Injector ................................. 5
1.5 Overview of Previous Research ........................................ 7
1.6 Research Scope ........................................................ 10
  1.6.1 Objective and Approach ......................................... 10
  1.6.2 Overview of the Present Study .................................... 11

2 Injector Design Methodology ............................................. 13

2.1 Overview of Injectors Investigated .................................... 13
CONTENTS

2.1.1 Geometric Design Parameters ........................................... 13
2.1.2 Injector Test Matrix .................................................. 14
2.2 Numerical Flowfield Description ....................................... 18
   2.2.1 Flow Conditions .................................................. 18
   2.2.2 Injector Geometry ................................................ 19
   2.2.3 Computational Domain
      Boundary Conditions ................................................. 22
      Convergence Criteria .............................................. 22
   2.2.4 Grid Generation .................................................. 23

3 Numerical Methodology ..................................................... 28
   3.1 Physical Modeling Issues ............................................ 28
      3.1.1 Overview of Numerical Algorithm ........................... 30
   3.2 Governing Equations ................................................ 30
   3.3 Temporal Discretization ............................................. 35
   3.4 Spatial Discretization: Shock-Capturing Scheme ................ 39
      3.4.1 Efficient Computational Forms ............................... 43
   3.5 Boundary Conditions ................................................ 45
      3.5.1 Inflow/Outflow Conditions .................................. 45
      3.5.2 Surface Boundary Condition ................................ 46
      3.5.3 Symmetry Condition ........................................... 47
   3.6 Curvilinear Transformations: Metrics .............................. 47
      3.6.1 Metric Discretization ....................................... 48
   3.7 Code Validation ..................................................... 50
      3.7.1 Flat Plate and Plane ......................................... 51
      3.7.2 Mach 2 Flat Plate ............................................ 51
      3.7.3 2-D 15° Hypersonic Compression Corner .................. 52
      3.7.4 24° Hypersonic Compression Corner ......................... 53

4 Numerical Injector Flowfield Results .................................. 63
   4.1 Analysis of Numerical Results ..................................... 63
      4.1.1 Flow Field Analysis .......................................... 63
      4.1.2 Performance Parameters
         Circulation ..................................................... 64
         Mixing Efficiency ............................................... 64
         Mixedness Measure ............................................. 65
Roman Symbols

\( a \) \hspace{1em} \text{acoustic speed}
\( A \) \hspace{1em} \text{Jacobian of } \xi\text{-direction inviscid flux vector } E
\( B \) \hspace{1em} \text{Jacobian of } \eta\text{-direction inviscid flux vector } F
\( C \) \hspace{1em} \text{Jacobian of } \zeta\text{-direction inviscid flux vector } G
\( C_p \) \hspace{1em} \text{specific heat at constant pressure}
\( e \) \hspace{1em} \text{specific internal energy}
\( f \) \hspace{1em} \text{stoichiometric fuel/air mass ratio}
\( E \) \hspace{1em} \text{total energy}
\( E \) \hspace{1em} \text{inviscid flux vector in } \xi\text{-direction}
\( E_v \) \hspace{1em} \text{viscous flux vector in } \xi\text{-direction}
\( F \) \hspace{1em} \text{inviscid flux vector in } \eta\text{-direction}
\( F_v \) \hspace{1em} \text{viscous flux vector in } \eta\text{-direction}
\( G \) \hspace{1em} \text{inviscid flux vector in } \zeta\text{-direction}
\( G_v \) \hspace{1em} \text{viscous flux vector in } \zeta\text{-direction}
\( h \) \hspace{1em} \text{specific enthalpy}
\( h_a \) \hspace{1em} \text{total specific enthalpy}
\( h_{k_i} \) \hspace{1em} \text{specific enthalpy of species } k_i
\( h_{a,k_i} \) \hspace{1em} \text{total specific enthalpy of species } k_i
\( I \) \hspace{1em} \text{identity Matrix}
\( J \) \hspace{1em} \text{metric Jacobian}
\( M \) \hspace{1em} \text{Mach number}
\( \mathcal{M}_{k_i} \) \hspace{1em} \text{molecular weight of species } k_i
\( P \) \hspace{1em} \text{pressure}
\( q_v \) \hspace{1em} \text{square root of cartesian velocity vector dot product}
NOMENCLATURE

\( \mathbf{Q} \) column vector of conservative variables
\( \mathbf{R}_c \) universal gas constant
\( \mathbf{R} \) residual
\( \mathbf{R} \) gas constant for a specific gas
\( \mathbf{R}_{\xi} \) matrix of right eigenvectors in the \( \xi \)-direction
\( \mathbf{R}_{\xi}^{-1} \) matrix of left eigenvectors in the \( \xi \)-direction
\( \mathbf{S} \) column vector of source terms
\( t \) time
\( T \) temperature
\( \mathbf{u} \) column vector of velocity components
\( u \) velocity component in the \( x \)-direction
\( U \) contravariant velocity in the \( \xi \)-direction
\( v \) velocity component in the \( y \)-direction
\( V \) contravariant velocity in the \( \eta \)-direction
\( w \) velocity component in the \( z \)-direction
\( W \) contravariant velocity in the \( \zeta \)-direction
\( x \) coordinate of physical reference frame
\( X_{k_s} \) mole fraction for species \( k_s \)
\( y \) coordinate of physical reference frame
\( Y_{k_s} \) mass fraction for species \( k_s \)
\( z \) coordinate of physical reference frame

Greek Symbols

\( \alpha \) injector geometry angle
\( \gamma \) specific heat ratio
\( \delta \) semiangle of cone or wedge
\( \varepsilon \) oblique shock wave angle
\( \zeta \) coordinate of computational reference frame
\( \eta \) coordinate of computational reference frame
\( \vartheta \) grid clustering parameter
\( \kappa \) thermal conductivity
\( \lambda(\mathbf{A}) \) eigenvalues of flux vector Jacobian \( \mathbf{A} \)
\( \mu \) dynamic viscosity
\( \xi \) coordinate of computational reference frame
\( \rho \) density
\( \rho_{k_s} \) density of species \( k_s \)
\( \tau_{r,s} \) shear stress
\( \varphi \) fuel/air equivalence ratio
Superscripts and Subscripts

- $\circ$ total or stagnation property
- $a$ air (used with flow properties)
- $c$ compression (used with $\alpha$)
- $e$ expansion (used with $\alpha$)
- $f$ fuel (used with flow properties)
- $i$ first spatial index
- $j$ second spatial index
- $k$ third spatial index
- $k_s$ species index
- $n$ index of time step or iteration
- $N$ total number of governing equations
- $N_s$ total number of chemical species
- $\infty$ freestream conditions
- $s$ sweep (used with $\alpha$)
1.1 Background and Motivation

The doubtless future increase in space exploration, high speed transport and demand for satellite deployment will necessitate advances in hypersonic propulsion technologies which hold the promise of improved efficiency, reliability and cost reduction. Focus has been shifting from expensive and heavy rocket based propulsion for space delivery systems to air-breathing propulsion which includes enabling technologies for the achievement of future goals in an economically viable manner. These concepts utilize the air freely available in the atmosphere rather than carrying oxidizer on-board, which has an associated weight penalty, to achieve hypersonic speeds ($M > 7$). The mass fraction previously occupied by the oxidizer can now be used for a variety of functions including increased payload, redundant safety systems and passenger capacity. Of the air-breathing propulsion concepts envisioned, perhaps none capture the imagination more than supersonic combustion ramjets (scramjets), and the closely related, shock-induced combustion ramjet (shcramjet). Spaceplanes utilizing these modes of hypersonic propulsion furnish a host of exciting possibilities. The early 1990's witnessed a surge in interest in hypersonic air-breathing propulsion which spawned research into scramjet based spaceplanes capable of everything from hypervelocity terrestrial flights to affordable space launch and even space tourism.\textsuperscript{1-3} The technological challenges of air-breathing hypersonic propulsion, however, are great and many; yet the considerable promise of these hypersonic concepts continues to attract research interest. Of the challenges which impede concepts such as the shcramjet and scramjet from coming to fruition, perhaps the most crucial is the efficient
mixing of fuel and air. In order for the chemical reactions of combustion to proceed, the individual fuel and oxidizer molecules must come into intimate contact. Thus, some form of mixing is required between the airstream and the introduced fuel. Incomplete mixing leads to inefficient combustion and unacceptable losses in thrust performance. Given the direct impact fuel/air mixing has on combustion, and the integrated nature of hypersonic vehicles with their propulsion systems, all aspects of vehicle performance are largely dependent on mixing efficacy.

Mixing of fuel and air in the hypervelocity, high-enthalpy environment of hypersonic propulsion devices is a difficult task. In order to obtain hypersonic flight Mach numbers with ramjets, oncoming flow cannot be decelerated to subsonic velocities due to extreme temperature and pressure increase and prohibitive flow losses. These losses can be manifest in normal shock wave systems and combustion conditions which lose a considerable fraction of available energy to molecular dissociation. Consequently, incoming flow must rather be compressed to supersonic velocities, on the order of several thousand meters per second, for combustion. The necessity of supersonic combustion generates a difficult fuel/air mixing and combustion problem due to the short residence times in the combustor. A primary design criterion for hypersonic vehicles is to limit the length of the combustor. While this reduces the combustor cooling loads and weight, it further decreases the flow residence time in the combustor. Consequently, fuel and air must be mixed and burned in times on the order of milliseconds. Further hindering the fuel/air mixing process is the desire for near parallel injection of the fuel. This is done primarily to recover the momentum of the injected fuel for vehicle thrust and to mitigate losses. In the case of shock-induced combustion ramjets, where it is required to mix the fuel and air before combustion, parallel injection is also necessary so that the fuel does not ignite prematurely. This constraint further inhibits effective mixing because of the generally low fuel penetration due to poor shear layer mixing associated with simple parallel injection at high Mach numbers. Previous experimental and numerical studies have found that shear layer mixing becomes more ineffective as the compressibility of the flow increases.\textsuperscript{4,5} The primary fuel used for hypersonic propulsion is hydrogen (H\textsubscript{2}). While the hydrogen fuel contains ample amounts of chemical energy to provide the required heat release and vehicle thrust, it has a very low molecular weight which inhibits rapid penetration and dispersion.

Effective and rapid fuel/air mixing enhancement mechanisms and devices for parallel injection must clearly be sought for efficient combustion and maximum performance of hypersonic propulsion systems.
1.2 Mixing Enhancement Strategies

Mixing enhancement strategies are sought to improve, or augment, the mixing of fuel and air over simple parallel injection relying on shear-layer mixing. Mixing augmentation usually has an associated total pressure loss, however, which must be weighed against the gain in performance.

Mixing enhancement can be achieved by judicious design of devices which introduce the fuel into the airstream. These fuel injection devices, or injectors, can be classified into two main categories: non-intrusive injectors such as wall orifices, and those which employ a mechanical structure which protrudes into the main flow, such as struts and ramp injectors. These categories are not mutually exclusive or exhaustive but provide a suitable framework for an introduction to the most commonly utilized mixing enhancement techniques.

1.2.1 Non-Intrusive Injectors: Orifices

The first class of injectors primarily consists of orifices mounted flush to the combustor wall which inject the fuel at a given angle to the wall. Included in this class of injectors are low-angled fuel injection,\(^6\)–\(^{12}\) normal injection,\(^{13}\)–\(^{15}\) and even fuel injection upstream\(^{16}\) into the main flow. Wall injection can be from circular or non-circular orifices\(^{17},^{18}\) and an array of orifices\(^{19}\)–\(^{21}\) can be constructed of different sizes and shapes to enhance mixing. The primary drawback of flush fuel injectors at low angles is insufficient penetration, while at higher angles, large losses become problematic. They are also more suited to applications which simultaneously mix and burn fuel due to probable ignition at the injection site.

1.2.2 Strut Injectors

Injection struts have a wide range of geometries and configurations.\(^{22}\)–\(^{25}\) Their advantage lies in the fact that the fuel can be introduced into the freestream far from the low-momentum, high temperature wall regions of a combustor or propulsive duct. Typically struts are constructed with a slender aerodynamic cross-section, which minimizes drag and flow blockage, and fuel is issued from ports along the downstream face of the strut. Appreciable losses, cooling requirements and materials considerations limit their applicability for high enthalpy flows however.

1.2.3 Ramp Injectors

A fuel injection system that has received growing interest is the ramp injector.\(^{26}\)–\(^{37}\) Ramp injectors are either mounted on a wall surface or become an integral part of the wall contour
and inject fuel at low angles or parallel to the surrounding airflow from a downstream facing step or ramp. These injectors provide for fuel momentum recovery and generally lower losses compared to normal injection. A variety of intrusive fuel injector designs have been proposed but the most promising of these devices are those which employ axial vortices to mix the fuel and air. These vortices have been shown to be effective for high Mach number flows which makes them ideally suited for hypersonic mixing applications. Ramp fuel injectors which exploit axial vortex generation techniques show great potential for enhanced fuel/air mixing.

The aim of these injectors is to convert a small fraction of the freestream momentum into angular momentum to produce axial vortices in the interest of mixing enhancement. A ramp injector design which employs axial vortices is the so-called "conventional" ramp fuel injector, shown schematically in Fig. 1.1a. This type of fuel injector was first introduced by Marble et al. and has subsequently been the focus of considerable research. This type of injector is the predecessor of a derived class of injectors investigated in the present thesis study. A brief review of axial vortex generation mechanisms is appropriate before the introduction of the ramp injectors considered herein.

1.3 Axial Vortex Generation Mechanisms

Three dimensional vortex structures can be generated via a gas dynamic process when a pressure gradient is imposed upon a non-parallel density gradient. This process is caused by baroclinic torque which is shown in the vorticity equation as

\[ \frac{D}{Dt} \left( \frac{\omega}{\rho} \right) = \frac{1}{\rho^2} \nabla \rho \times \nabla p + \ldots \]  

where \( \omega \) is the vorticity vector. From Eq. 1.1 it can be seen that cross product implies vorticity generation only for non-parallel gradients. The physical interpretation of baroclinic torque can be viewed as the misalignment of the center of mass of a fluid particle with the line of action of the pressure forces on this fluid element resulting in a torque. The shock waves present in supersonic flow together with the density gradient produced by two non-like gases (fuel and air) provide the conditions necessary to practically implement baroclinic torque for mixing enhancement purposes. It has been shown that substantial vortex generation can be achieved by the deposition of baroclinic torque on a fuel/air interface when a column of light gas (such as hydrogen fuel) in a surrounding airflow passes through a shock wave. The effect of these vortices is the entrainment of the heavier gas into the lighter gas. Baroclinic torque is particularly appropriate for mixing applications since the vorticity is produced directly at the
fuel/air interface where mixing is desired.

Other axial vortex generation mechanisms are produced in the context of ramp fuel injectors. The three dimensional geometries of these injectors provide gas dynamic conditions conducive to the generation of vortices. These aspects will be discussed further, after a description of cantilevered ramp fuel injectors, in Section 1.4.

1.4 The Cantilevered Ramp Fuel Injector

In an effort to improve the mixing performance of the conventional ramp injector, a new modified version of this injector was developed at the University of Toronto Institute for Aerospace Studies (UTIAS) and dubbed the "cantilevered" ramp injector (Fig. 1.1b). This type of ramp injector forms the focus of the present thesis study which constitutes the very first investigation of the cantilevered design. Some of the salient features of the cantilevered ramp injector are detailed in Fig. 1.2. This injector concept is thought to embody characteristics of the conventional ramp injector and low-angle wall fuel injection. It is derived from a conventional ramp injector and thus has several features in common with it. An injector array consists of alternate compression ramps and expansion troughs (see Fig. 1.1) which are characterized by angles $\alpha_c$ and $\alpha_e$ (see Fig. 1.2). These angles produce shock structures as presented schematically in Fig. 1.2. Fuel is injected from the end of each ramp into the surrounding airflow. It can be seen that the fuel introduced by the injector passes through a shock wave shortly downstream of injection. This situation sets the stage for baroclinic torque vortex generation with the fuel/air contact surface providing the density gradient and the shock supplying the pressure gradient. The second major axial vortex generating mechanism provided by the ramp fuel in-
jector results from the high-pressure fluid from the upper surfaces of the ramps spilling to the low-pressure regions (troughs) between the ramps (see Fig. 1.2b). This mechanism is referred to as cross-stream shear.

The distinguishing feature of the cantilevered ramp injector is that the section $BO'B'$ is removed which allows air flow under the injected fuel. This key design change has the potential for improved fuel/air mixing. In addition to the side wall vortices generated by the cross-stream shear, strong vortices will be produced behind the "bluff-body" of the injector (vortices $M$ and $N$ in Fig. 1.2b), as in the case of a low-angle wall fuel injector. These vortices are generated in close proximity to the injected fuel which has the potential to further enhance the mixing process. The volume created under the cantilevered fuel injector also allows the formation of a more contiguous shock structure (Shock A) in the spanwise direction of the injector array. This increases the baroclinic effect and hence the large scale near-field mixing efficiency due to the increased pressure gradient at the fuel/air interface.
Although it can be considered as a candidate for fuel injection in scramjet combustors, the newly designed cantilevered ramp injector is primarily considered for use in shock-induced combustion ramjets, where fuel/air mixing should take place without combustion until a specific location in the propulsive duct of the engine. Unlike the low-angled wall fuel injection case, where a shock generated by the inclined high-momentum fuel jet would ignite the fuel/air mixture in the vicinity of the injection point, the cantilevered ramp completely envelopes and shields the fuel jet from the high-temperature region existing in the vicinity of the injector. Further, the cantilevered design has the inherent advantage of an elevated injection plane remote from the high-temperature lower wall region. This improvement over the conventional ramp design makes the cantilevered ramp injector well suited to shock-induced combustion ramjets.

Finally, the cantilevered design also has the advantage that the fuel injection plane can be moved closer to the shock structure primarily responsible for baroclinic torque vortex generation. It can be seen from Fig. 1.2a that the tip of the fuel injector can be extended downstream such that the fuel injection plane \((c - c)\) translates from point \(B'\) to point \(B\). Since Shock \(A\) is still initiated at point \(B''\), the resulting modification results in earlier deposition of baroclinic torque on the injected fuel.

## 1.5 Overview of Previous Research

Ramped fuel injectors employed as a means of fuel/air mixing enhancement has been the subject of a considerable amount of previous research. Interest in these fuel injectors was largely initiated by the National Aero-Space Plane (NASP) program in an effort to improve fuel/air mixing in scramjets.

In an early experimental effort, Northam et al. investigated a variety of wall-mounted injector ramps.\(^{27}\) Both swept and unswept ramp injectors were studied in various duct configurations. Emphasis was placed on near parallel injection for thrust recovery at high vehicle flight Mach numbers. Consequently, fuel was injected from ramps at an angle of 10.3° to the combustor wall. Fuel at \(M_f = 1.7\) was injected into a \(M_a = 2\) airflow. The injector design incorporated reflected shock waves intersecting the injected fuel to enhance mixing. Analysis of results comprised largely of shadowgraph flow visualization and combustion efficiency calculations. Results found the swept ramp injector to have generally superior performance over the unswept ramp. A combination of ramp and subsequent downstream perpendicular injection was found to improve combustion efficiency only in the case of unswept injectors.

Furthering the research of Northam et al., an experimental effort was conducted by Hartfield et al. considering very similar ramp injectors.\(^{28}\) Their work revealed highly three-dimensional
flow fields which “dramatically illustrate the domination of the mixing process by streamwise vorticity generated by the ramp”. Experiments consisted of non-reacting flow with seeded air injected into freestream air for freestream conditions of Mach 2 and Mach 2.9. A primary objective of the study was the determination of the influence of freestream Mach number on injector performance for a given injector geometry. A laser-induced iodine fluorescence technique was employed to collect temperature and injectant concentration data. Analysis of global mixing performance was limited to a parameter reflecting the percentage of duct area mixed to within static flammability limits. It was found that the injectant mixed faster at lower freestream Mach numbers. Comprehensive flow field visualization was presented clearly delimiting the vortical flow structures.

Complementing the previous experimental studies was a numerical investigation by Riggins et al. A reacting Navier-Stokes code was utilized to solve both swept and unswept ramp configurations with fuel injected at Mach 1.7 through circular orifices from ramp injectors. The code employed a two step finite-rate chemistry model together with the Baldwin-Lomax turbulence model. Both reacting and non-reacting cases were considered with laminar flow calculated for non-reacting cases. The numerical solutions afforded a more thorough performance analysis including measures of circulation, fuel concentrations, mixing efficiency, total pressure recovery and, in the case of reacting flow, combustion efficiency. Basic gridding techniques and boundary condition treatments were used and grid convergence issues were not addressed. Results showed substantially higher mixing performance as well as flow losses for the swept configuration over the unswept ramp. The study concluded that vorticity increased fuel mixing and that near-field mixing was controlled by large-scale vortices while far-field mixing was controlled by smaller scale turbulent diffusive processes.

A significantly improved injector design was proposed by Marble et al. who introduced the “contoured wall fuel injector”. The ramp injector design was integrated with the combustor wall which allowed for shock structures promoting baroclinic torque generated vorticity. A combined computational and experimental effort focused on the demonstration of enhanced mixing through the generation of streamwise vorticity and its use for hypersonic propulsion. It was determined that the characteristic mixing times were fast enough for scramjet applications. While limited quantitative analysis and no comparison with other designs was presented, a ground breaking proof of concept was shown. The investigation concluded that the ramp injector under consideration “can lead to rapid enhancement of the mixing process”. A further conclusion was that a mechanism to destabilize the large vortices must be sought to ensure complete mixing.

A modified design to Marble’s injector was studied by Davis et al. The fuel injector em-
bodied the elements of the contoured wall fuel injector but was more modular since it could be mounted on a flush combustor wall. The study focused on jet penetration and mixing behavior under a variety of different operating conditions. A swept and unswept injector configuration was experimentally tested. Unfortunately, quantification of mixing performance was minimal and comparison with other injectors was absent. The superior vortex generating ability of the swept configuration over the unswept was established.

A thorough investigation of the injector design advanced by Marble et al. was undertaken by Waitz et al.\textsuperscript{42,43,46} A concurrent experimental and numerical effort was undertaken to study Mach 1.7 helium (used to simulate hydrogen) injected into a Mach 6 airstream. Several parametric dependencies were investigated including: injector spacing, ramp geometry, wall boundary layer effects and injectant/freestream velocity and pressure ratios. A detailed description of flow fields and flow phenomena were presented. The work demonstrated that the induced vorticity coalesced into a counter-rotating pair of vortices promoting helium migration up into the main stream. The two main sources of vorticity, baroclinic torque and cross-stream shear, were identified and characterized. It concluded that shock-impingement produced effective mixing by deposition of baroclinic torque at the fuel/air interface while cross-stream shear induced vorticity can be less effective due to vortices generated remote from the fuel/air interface. Flow visualization was employed to identify salient flow features. Excellent comparisons of experimental and computational results were presented along with comprehensive mixing performance and loss analysis. The suitability of injector design to scramjet applications was addressed and it was concluded that the injector design in question was a feasible candidate for mixing enhancement.

An interesting approach to improve fuel/air mixing enhancement is through the use of various nozzle geometries used to inject fuel from ramp injectors.\textsuperscript{29} Haimovitch et al. experimentally investigated six different injector-nozzle inserts to precondition the fuel flow. The main objective was to determine the influence of the resultant fuel jet on the vortical flow field induced by the ramp injectors. Seeded air at Mach 1.63 was used to simulate fuel injected into a Mach 2 main stream. Mie scattering visualization revealed only minor difference in the mixing performance between the candidate injectors.

More comprehensive computational results were provided by Eklund et al. who studied mixing in the context of a reacting flow field.\textsuperscript{31} The Navier-Stokes equations were solved with a finite-rate chemical kinetics model for H\textsubscript{2}-air reaction together with an algebraic eddy viscosity turbulence model. Experiments using high-enthalpy air were conducted for comparison with computations. Two configurations: swept compression and swept expansion ramp injectors, were used to inject Mach 1.7 fuel at 10° with respect to the main flow. A major conclusion of
the investigation was that mixing was significantly reduced by combustion. A reduction of up to 25% in mixing efficiency was observed for the reacting case.

Riggins and Vitt furthered investigation of generic swept and unswept ramp injectors with a more refined numerical model.26 Larger computational grids were employed simulating laminar and turbulent mixing. Insightful analysis of results underscored the dominant role of turbulence in the far-field, although turbulence modeling issues were not fully addressed. Comparison with high-enthalpy experimental results determined that CFD is a valuable engineering tool for injector design.

Still more sophisticated numerical modeling was applied to ramp injectors by Lee et al.47 The contoured wall injector design of Marble was investigated to determine the mixing characteristics in the presence of combustion. A numerical algorithm employing the three-dimensional Navier-Stokes equations coupled with a chemical reaction model and a $k - \varepsilon$ turbulence model was used for the study. Freestream air conditions were held constant while changing initial fuel pressure and density. The study concluded that changes in fuel density had a significant impact on mixing and combustion performance while pressure changes had little effect. It further asserted that the mixing process has a strong influence on combustion, whereas the combustion process does not have any significant effect on the mixing process. The results suggest that the mixing process may be decoupled from the combustion process with only minor differences in performance trends.

A more intricate ramp injector configuration was studied by Baurle et al. in a combined experimental and computational investigation.36 Ramp injectors were mounted on opposite sides of the combustor in an interdigitated fashion with four fuel injection ports located at the base of each ramp. The nozzles in the base of the ramps were angled with respect to the combustor wall and a yaw angle was also introduced. The injector nozzle flow was included as part of the computational domain which consisted of a remarkable 13.5 million grid nodes. Comparison with experimental results was very favourable. The ramp injectors were compared with an equally innovative strut design in the context of mixing performance and total pressure losses. Overall, the particular strut injector considered showed marginal performance gains.

1.6 Research Scope

1.6.1 Objective and Approach

The main objective of the present study was to investigate the proposed cantilevered ramp injection systems to improve the fuel/air mixing process in airflows representative of hypersonic
vehicle conditions. This thesis represents the very first study of this new injector design.

Within the context of the cantilevered ramp injector, there are many avenues of potential research. It was desired that the viability of the new cantilevered ramp injector design be determined by comparison with conventional ramp injector designs. Further, germane aspects of the cantilevered injector design were sought from a parametric study such that design criteria could be established for improved performance in subsequent designs. Understanding of the role of the various vortex generation mechanisms on mixing enhancement was sought. The influence of various geometric parameters on mixing performance was also desired. Therefore, the body of research within the scope of the present thesis study was the investigation of two main aspects:

1) Comparison of the mixing performance characteristics of the proposed cantilevered ramp designs with conventional designs, and

2) Investigation of the cantilevered ramp fuel injectors in a parametric study of various geometries.

A computational effort was undertaken to achieve the aforementioned objectives. A three-dimensional Navier-Stokes algorithm was developed and implemented to solve the flow fields of both conventional and cantilevered ramp injectors. Ramp fuel injectors were designed with various geometrical parameters while holding the flow conditions constant. Comparison of the resultant flow fields provided insight into the mixing performance of various cantilevered ramp injectors. Quantitative comparison of injector performance was possible through a number of bulk mixing parameters providing insight into mixing efficacy as well as flow losses.

1.6.2 Overview of the Present Study

Both conventional and cantilevered ramp injectors were designed for subsequent numerical solution. The complete design methodology of the fuel injectors investigated is presented in Chapter 2. Elements of injector design include: the geometric parameters which describe the physical surfaces of the injector, the flow conditions considered, and aspects of the design of the numerical flow field used to simulate the injector. The development of the numerical algorithm employed to solve the injector flow fields is outlined in Chapter 3. The three-dimensional laminar Navier-Stokes equations are implemented for the present study. Details of the temporal and spatial discretization processes are presented together with salient aspects of the numerical model used. The numerical solutions of the injector flow fields are given in Chapter 4. Analysis of the flow structures and performance parameters of the various injector configurations are
discussed and conclusions drawn. Closure of the present study is provided by Chapter 5 which presents design recommendations and suggestions for future study.
The design issues associated with the complete description of the fuel injectors considered herein is detailed, including the numerical aspects which constitute the complete formulation of a computational simulation. The physical injectors, including their geometry, are first addressed followed by modeling details required for solution employing the numerical approach outlined in Chapter 3.

2.1 Overview of Injectors Investigated

The fuel injectors considered in the present study can all be broadly classified as ramp-type fuel injectors. The focus of investigation was directed toward the "cantilevered" ramp injector. The "conventional" ramp injector is considered for comparison with the cantilevered design. These two types of injector are shown in Fig. 2.1. Many configurations of these fuel injectors are possible by specification of various geometric parameters. These are set out in the following sections.

2.1.1 Geometric Design Parameters

The defining geometric parameters of the conventional and cantilevered ramp injector configurations considered are shown schematically in Figs. 2.2 and 2.3. The compression and expansion angles are denoted by $\alpha_c$ and $\alpha_e$ respectively. These angles are responsible for the strength of the shock waves which, in turn, largely govern the axial vortex generation mechanisms re-
viewed in Section 1.3. The overall length of a given injector is defined as the distance between the lip of the compression ramp (or expansion trough) to the downstream tip of the injector (corresponding to the streamwise location of fuel injection) and is denoted by \( L \). The height and width of the fuel injection plane are defined as \( h \) and \( w \). For the case of the rectangular cross sections considered, the simple area relation, \( A_f = h \times w \), is valid and notationally consistent for all injector configurations. The angle \( \alpha \), denotes the angle of sweep of the side walls of the injector. Sweep angle is often introduced to increase the vorticity via enhanced cross-stream shear flow. The extra geometric variable in the case of a cantilevered ramp injector is the vertical spacing below the injector; this is shown as \( \Delta z_c \) in Fig. 2.3. The number and spacing of fuel injectors completes the geometric description of a given configuration. Considering an infinite array of fuel injectors leads to two symmetry planes, which are labeled \( S_1 \) and \( S_2 \). The first symmetry plane represents the symmetric nature of an individual injector unit while the second, \( S_2 \), follows from equal spacing of injectors in the spanwise direction. The distance between symmetry planes is denoted by \( \Delta y \).

### 2.1.2 Injector Test Matrix

A comprehensive numerical test matrix populated by the permutations of the aforementioned geometric parameters would indeed lead to a very large set of configurations which are impractical to solve numerically. Further parametric studies are possible with variations in either the freestream air properties, fuel inflow conditions, or both. In keeping with the objectives and scope of the current study, a select number of fuel injector configurations was considered.
FIGURE 2.2: Geometric parameters for conventional ramp fuel injector.

FIGURE 2.3: Geometric parameters for cantilevered ramp fuel injector.
The first simplification of a fully populated test matrix was to impose constant fuel and air inflow conditions. Thus, all injector configurations considered were numerically solved with the flow values presented in Section 2.2.1.

Of the geometric parameters which define the ramp fuel injector, focus was on the effect of ramp angles ($\alpha_c$, $\alpha_e$ and $\alpha_t$) on the mixing flow field. It was hoped that a parametric study on the compression and expansion angles ($\alpha_c$ and $\alpha_e$) would lend insight into the main vortex generation mechanisms present. The compression angle is largely responsible for the cross-stream shear vortex generation while the expansion angle largely influences the baroclinic torque axial vortex production; therefore, variation of these angles can clarify the respective impact on mixing performance of these mechanisms. Previous studies have demonstrated a significant influence on mixing performance by specification of a non-zero sweep angle. Both swept ($\alpha_s \neq 0$) and unswept ($\alpha_s = 0$) injector configurations are considered in the present test matrix.

Referring to the injector test matrix in Table 2.1, five main configurations are identified: A, B, C, D and E.

The first, and most general, configuration (A) indicates a conventional ramp or cantilevered ramp fuel injector with non-zero compression and expansion angles ($\alpha_c \neq 0$ and $\alpha_e \neq 0$). This configuration incorporates both of the main axial vortex generation methods previously described. Within the configuration A classification, four sub-configurations are denoted by A1, A2, A3 and A4. Configurations A1 and A2 represent a cantilevered ramp and conventional ramp fuel injector with unswept ramp sides ($\alpha_t = 0$) respectively, while A3 and A4 represent a cantilevered and conventional injector with swept side walls ($\alpha_s \neq 0$).
Configuration B was constructed to accentuate the baroclinic torque and reduce the cross-stream shear component of axial vortex generation. This configuration consists of a zero compression angle and non-zero expansion angle ($\alpha_c = 0, \alpha_e \neq 0$). Configurations B1 and B2 represent unswept cantilevered and conventional ramp fuel injectors respectively.

Conversely, configuration C was devised to shift emphasis onto the cross-stream shear axial vortex generation mechanism. This is accomplished by specifying $\alpha_c \neq 0$ and $\alpha_e = 0$. Configurations C1 and C2 represent unswept cantilevered and conventional ramp fuel injectors.

Configuration D utilizes injector geometries situated on opposite walls of a mixing duct, staggered in the spanwise direction. This configuration provides for the study of the interaction of multiple injector flow fields and their potential for more effective mixing performance. Configuration D1 denotes an unswept cantilevered ramp injector with non-zero compression and expansion angles ($\alpha_c = 0, \alpha_e \neq 0$ and $\alpha_e \neq 0$).

The cantilevered ramp injector provides the potential to translate the fuel injection plane further downstream. Specifically, the streamwise location of the injection plane must not necessarily be coincident with the downstream extent of the expansion trough; the fuel injector tip can effectively be extended further downstream. This configuration is denoted by configuration E in Table. 2.1. This fuel injector has the advantage of earlier deposition of baroclinic torque and was designed to address the possibility of cantilevered ramp design optimization.

Two additional studies undertaken on the above injector configurations are: 1) an incident boundary layer study and, 2) a far-field study. These studies are appended as the final two rows of the injector test matrix (Table. 2.1).

The incident boundary layer study investigates the effect of a viscous boundary layer profile on the solid surfaces upstream of a fuel injector. This is accomplished by solution of a constant area duct portion upstream of the fuel injector geometry and using the resulting profile as inflow to the injector sub-domain (see Section. 2.2.3). The designation BL-A1, for example, indicates the effects of an incident boundary layer was studied for the A1 injector configuration (see above).

The far-field study was undertaken to address the mixing performance at significant distances downstream of fuel injection. This was accomplished by extension of the computational domain further downstream (see Section. 2.2.3). The designation FF represents the far-field study with an extended designation (eg. FF-A1) referring to a case of a specific injector configuration.

The specifics of the aforementioned configurations are detailed in Section 2.2.2 and summarized in Table. 4.1.
2.2 Numerical Flowfield Description

Before undertaking a numerical simulation of the flow around a given fuel injector, a detailed description of the flow field is required for the flowsolver. The present section provides the specific information necessary to completely describe the numerical flow field, including: the flow conditions, injector geometry, computational domain and the grid generation details.

2.2.1 Flow Conditions

The fuel and air entering the computational domain were determined from the expected conditions of the hypersonic propulsion concepts in which the injectors are envisioned to be utilized. The air inflow conditions used to numerically model both the conventional ramp and cantilevered ramp injectors were chosen to be representative of the high Mach number, high enthalpy shock-induced combustion ramjet combustor environment, corresponding to a flight Mach number of $M_{\infty} = 14.51$. The fuel used in the current study is hydrogen (H$_2$) since it is the most commonly used fuel in the propulsion concepts of interest.

The cryogenic fuel used in many hypersonic propulsion devices such as the shock-induced combustion ramjet, will be used as an airframe coolant. Thus, it is expected that the stagnation temperature of the fuel prior to injection be relatively high. For this reason a value of $T_{c,f} = 850$ K was chosen for the total fuel temperature at the injector inflow. Further, to limit turbulence modeling issues, the velocity of the fuel and air were matched at their respective inflow boundaries. Since the focus of the present study is consideration of mixing enhancement via gas dynamic mechanisms, such as baroclinic torque, turbulence modeling is not required or considered. Pressure at the inflow boundaries of the fuel and air were matched in order to avoid excessive expansion or compression. These three constraints completely define the thermodynamic state of the hydrogen at the injection plane. No effort was made to model the fuel injector nozzle flow; fuel inflow conditions were specified as uniform and constant over the injection plane.

The inflow air and hydrogen conditions were kept constant for all cases considered and are summarized in Table 2.2. Fixing the inflow conditions of the fuel and air, together with a constant equivalence ratio, facilitates ease of comparison among configurations since the same mass flow rate of fuel and air is introduced into the domain of each injector considered. Both the fuel and air inflow were modeled as parallel to the streamwise axis.
Table 2.2: Air and fuel freestream conditions.

### 2.2.2 Injector Geometry

The surfaces of an injector geometry define the solid boundaries of the computational domain which largely determine the numerical flow field solution. The design specifics of the ramp fuel injectors used in the computer simulations are elucidated in the present section.

The input parameters used to define a particular injector configuration are (see Fig. 2.2 and 2.3): 1) the height and width of the fuel injection plane \((h \text{ and } w)\); 2) the compression and expansion angles \((\alpha_c \text{ and } \alpha_e)\); 3) the sweep angle \((\alpha_s)\); 4) the distance between symmetry planes \((\Delta y_e)\); and, in the case of a cantilevered ramp injector, 5) the height of the injector tip above the lower wall \((\Delta z_c)\). The length of the injector is uniquely defined by specification of the above parameters. Simple geometrical considerations between the above variables completely define the injector surfaces. The number of geometric parameters opens a potentially very large parametric space available for study. For the present investigation, only a select few of these parameters are varied while the remaining are held constant for all configurations.

In choosing the injector geometry parameters, a number of considerations are relevant. As one of the main aims of the current work is the mixing performance comparison between the proposed cantilevered ramp injector and the conventional ramp injector, care was taken to ensure matching of the geometry and scale of corresponding configurations wherever possible. The size of the injectors must be chosen such that a numerical solution is tractable. Excessively large injectors lead to unmanageable computer memory and speed requirements. The height and width of the fuel injection plane was chosen to be 20 mm \(\times\) 20 mm for the cantilevered fuel injector and 10 mm \(\times\) 40 mm for the conventional ramp fuel injector. These parameters are held constant for all conventional and cantilevered configurations respectively. The projected area of the fuel injection plane is common for all injectors considered; \(A_f = 400 \text{ mm}^2\). The value, \(\sqrt{A_f} = 20 \text{ mm}\), is used as a non-dimensionalization parameter for length scales (eg. \(\bar{x} = x/\sqrt{A_f}\)) subsequently presented in the numerical flow field results. The decision to specify...
a square cross-sectional area for the cantilevered ramp injector and a rectangular cross-section \((h = 4w)\) for the conventional ramp injector was influenced by the criteria of matching the overall scale as well as the mass of fuel introduced by both injectors. Imposing the same injector cross-section for the cantilevered ramp injector as the conventional injector would lead to prohibitively large injector lengths (due to the introduction of the space under the injector, i.e. \(\Delta z_c \neq 0\)). This size discrepancy would complicate gridding issues and limit meaningful comparison between injector types.

The compression and expansion angles of a ramp injector are two of its primary characteristics. Larger angles are likely to produce stronger shock structures, greater vortex generation and thus better mixing performance. This increase in performance, however, comes at the cost of increased flow losses. These angles should be chosen large enough to produce the vortex structures of interest but not too large as to incur unacceptable losses. Indeed, it is reasonable to assume that a set of optimal angles exist, however, a parametric study on these angles is beyond the scope of the present investigation. The non-zero compression and expansion angle for all of the injectors considered was arbitrarily chosen to be seven and negative seven degrees respectively \((\alpha_c = 7^\circ, \alpha_e = -7^\circ)\). These angles were chosen by careful review of previous research and deemed to be reasonable for the present study. It is also noted that larger angles will produce shorter, more compact injectors.

For most of the injectors studied, the ramp sweep angle, \(\alpha_r\), was set to zero. In the case of the so-called swept injector configurations, an angle of five degrees was chosen (i.e. \(\alpha_r = 5^\circ\)).

Injector spacing can have significant effects on the flow field. Individual injector units must be located close enough together in order to deliver the requisite mass of fuel for a given area to achieve near stoichiometric proportions. Conversely, narrow spacing may produce undesirable interactions between injector units and minimize the flow in the expansion troughs.

The distance between symmetry planes was held constant for all injectors considered; \(\Delta y_z = 1.5 \sqrt{A_f} = 30\) mm.

The height of the computational domain is determined such that the requisite amount of air enters to produce a fuel/air equivalence ratio of \(\varphi = 1.1\). This equivalence ratio was specified with consideration to limited computational resources.

### 2.2.3 Computational Domain

The computational domain is formulated in the context of a multi-block grid system. A multi-block formulation has the advantage that each surface of a given block has only one applicable boundary condition and generation of individual blocks is simplified and more flexible given that clustering and cell density can be specified on a block basis. The number of computational
cells employed for a complete 3-D injector flow field is typically on the order of one million cells and can exceed two million depending on the configuration. In order to make the solution of such a large number of computational cells more tractable with available computer resources, the complete domain is segmented into a number of smaller multi-block sub-domains which are solved separately, in succession, with increasing streamwise distance. The outflow solution of the previous domain is specified as inflow conditions for the adjoining block system downstream. Each sub-domain is solved in a fully elliptic manner, however, there is no mechanism for upstream characteristic migration across sub-domain boundaries (ie. at the outflow boundary of a given sub-domain from the downstream sub-domain). Since, in the main, the outflow is supersonic, with the exception of boundary layers, the division of the overall domain has negligible affects on solution accuracy. As confirmation, a coarse grid injector domain was solved as a whole and in sub-domain segments with no meaningful difference in final solutions.

The general schematic of the sub-domain system used for a representative injector configuration is shown in Fig. 2.4. The primary injector sub-domain (labeled \( SD_1 \) in Fig. 2.4) extends from the start of the injector to ten reference lengths (\( \bar{x} = 10 \)) downstream of fuel injection while the secondary domain (labeled \( SD_2 \) in Fig. 2.4) extends from 10 to 40 reference lengths downstream of the injection plane. Sub-domain \( SD_0 \) is used for cases which require an initial viscous boundary layer profile as initial conditions to the injector while sub-domain \( SD_3 \) is employed for the "far-field" study outlined above; it extends from 40 to 70 reference lengths downstream of hydrogen injection. The orientation of the axes is such that the streamwise direction is aligned with the \( x \)-axis, the spanwise direction is measured by the \( y \)-axis and the extent above and below the injector is denoted with the \( z \)-axis, corresponding to a right-hand coordinate system. The origin of the entire domain, \( O, (\bar{x} = 0, \bar{y} = 0, \bar{z} = 0) \) is located at the fuel injection plane (in the streamwise direction) where the injector plane of symmetry and the lower solid surface intersect.

The number of blocks in each primary sub-domain (\( SD_1 \)), is dependent on injector configuration. A schematic of the block connectivity is given in Appendix A for various injector
configurations. Subsequent downstream sub-domains ($SD_2$ and $SD_3$) consist of a six block system which align with the corresponding grid systems at $\xi = \xi_{\text{min}}$ and $\xi = \xi_{\text{max}}$ where applicable.

**Boundary Conditions**

The boundary conditions driving the numerical simulations are implemented as described in Section 3.5 and consist of inflow, outflow, wall and symmetry conditions in the case of the fuel injector computational domains considered. The freestream flow impinging on the injectors is modeled as pure air ($N_2$ and $O_2$ in the molar ratio 3.73) while the flow issuing from the injectors is modeled as pure hydrogen ($H_2$) with the conditions set out in Section 2.2.1. At these computational boundaries the inflow condition is applied and thus a uniform flow is specified and held constant. The downstream streamwise boundary ($\xi = \xi_{\text{max}}$) is treated as an outflow boundary. All solid surfaces of the injector as well as the duct floor ($\zeta = \zeta_{\text{min}}$) and ceiling ($\zeta = \zeta_{\text{max}}$) were modeled with a wall condition with a specified temperature of $T_{\text{wall}} = 800$ K. This high temperature is indicative of a reasonable value which may be achieved by hypersonic vehicle cooling systems and the upper limit imposed by materials considerations. For computational efficiency, advantage was taken of all symmetry planes (see $S_1$ and $S_2$ in Figs. 2.2–2.3). Symmetry boundary conditions are implemented at the center plane of a given injector unit and at a plane between adjacent injectors ($\eta = \eta_{\text{min}}$ and $\eta = \eta_{\text{max}}$), thus modeling an infinite array of injectors in the spanwise direction.

**Convergence Criteria**

Iterative procedures, such as the solution of a set of non-linear partial differential equations transformed to algebraic equations, can be solved to increasing degrees of accuracy to the limit of the computer’s capability. Contemporary computers have the ability to routinely calculate values with 16 decimal place accuracy and beyond. Solving these equations to “machine zero” can entail large computer run times and is practically not required since, depending on the particular application, the variables solved are needed only to finite accuracy (four or five decimal places is usually sufficient for flow variables). In the case of three dimensional calculation of the Navier-Stokes equations, computational time for a given solution can be rather large and a degree of accuracy, or convergence criteria, must be established which is greater than machine zero to make the solution times reasonable.

The main variable monitored to assess the degree of convergence was the root mean square
of the relative density correction vector

\[ L = \sqrt{\frac{1}{N_c} \sum_{i=1}^{N_c} \left( \frac{\Delta \rho_i}{\rho_i} \right)^2} \]  

(2.1)

where \( N_c \) is the total number of cells in the computational domain. The convergence criterion adopted for the present work was the reduction of \( L \) to a minimum of five orders of magnitude from its initial value. This criterion was determined by monitoring various flow variables until there was no substantial change in the flow field and adding a margin of safety. The maximum density correction (the \( L_{\infty} \)-norm) was also monitored to ensure an acceptable degree of convergence.

### 2.2.4 Grid Generation

The computational grids used in the current work are of the structured variety, which is to say that the points are distributed along continuous, ordered lines. In order to conform to the irregular geometries dictated by the fuel injectors under study, a transformation of the three Cartesian spatial dimensions to a generalized set of coordinates was undertaken. This transformation provides a unique mapping of the nonrectangular grid in physical space to a uniform rectangular grid in the computational space. The generalized coordinates can be expressed as

\[ \xi = \xi(x,y,z) \]
\[ \eta = \eta(x,y,z) \]
\[ \zeta = \zeta(x,y,z) \]

(2.2)
(2.3)
(2.4)

and are derived in more detail in Section 3.6.

The computational domains for the fuel injectors are constructed in a multi-block formulation (ie. a series of smaller manageable blocks). The specification of these individual blocks follow a common methodology.

Firstly, the number of grid lines in each direction is specified along with the type and degree of clustering for each of the 12 cube edges for a three dimensional grid block. The clustering appropriate for the present application consists of three types: 1) grid concentration at the minimum domain boundary; 2) the maximum domain boundary or, 3) at both boundaries. Grid
Clustering is accomplished, for example, via the transformation\textsuperscript{53}

\[
    z = z_{\text{min}} + \left( z_{\text{max}} - z_{\text{min}} \right) \frac{(\beta + 1) - (\beta - 1) \left\{ \frac{[(\beta+1)]^{1-\zeta}}{[(\beta-1)]^{1-\zeta}} \right\}}{1 + 1}
\]

(2.5)

for clustering at a minimum domain boundary and the analogous relation for clustering at maximum domain boundary or at both boundaries given by\textsuperscript{53}

\[
    z = z_{\text{min}} + \left( z_{\text{max}} - z_{\text{min}} \right) \frac{(2\theta - \beta) + (2\theta + \beta) \left\{ \frac{[(\beta+1)]^{1-\zeta}}{[(\beta-1)]^{1-\zeta}} \right\}}{(2\theta + 1) \left\{ \frac{[(\beta+1)]^{1-\zeta}}{[(\beta-1)]^{1-\zeta}} \right\} + 1}
\]

(2.6)

where the parameter \( \theta \) is set to zero for clustering at the maximum boundary and \( \theta = 0.5 \) for concentrations at both boundaries. In both expressions the parameter \( \beta \) corresponds to the degree of clustering or grid cell concentration. It ranges \( 1 \leq \beta \leq \infty \) with increased clustering as \( \beta \to 1 \) and a uniform distribution as \( \beta \to \infty \). These functions allow for a smooth transition in node spacing which is more amenable to stable numerical convergence.

Secondly, given the node distribution on the cube edges, nodes on opposite boundaries of a given face are connected with straight lines to form a two-dimensional mesh on all cube surfaces.

Finally, nodes on opposite faces are interconnected with straight lines to complete the definition of a three-dimensional grid space.

Clustering at various locations is dictated by the expected physics contained within the numerical solution. Grid cells are concentrated near all solid surfaces such that the high gradients in the viscous boundary layers can be resolved to an acceptable degree of accuracy. Clustering is also specified at abrupt changes in geometry where flow discontinuities (shocks) are expected.

By insuring that surfaces of adjoining blocks have identical number of nodes and clustering, the assembly of the individual blocks form a continuous and consistent multi-block grid which fills the entire computational domain of interest. The numerical algorithm has been constructed in such a way that the flow field solution is independent of the location and number of individual block boundaries. In this respect the grid system is transparent to the final solution.

Figures 2.5 to 2.7 show portions of the computational domain for representative injector configurations. Fig. 2.5 illustrates a number of two-dimensional planes in the three-dimensional
domain for the primary sub-domain ($SD_1$, see Section 2.2.3) of the cantilevered ramp injector configuration. A cross-stream plane located just downstream of the fuel injection plane ($\bar{x} = 0$) is shown in Fig. 2.6. The mesh is mirrored about the $y$-axis in order to demonstrate the extent of an individual (symmetric) injector unit. Clustering used to resolve upstream solid surfaces is apparent which also serves to better resolve the initial contact surface downstream of fuel injection. Grid consistency is illustrated in Fig. 2.7 which compares a portion of the injector symmetry plane for a conventional ramp injector with its cantilevered counterpart. For comparison between conventional and cantilevered ramp injector cases, care was taken to construct the grid system such that the same grid density is present in corresponding regions of the domain.
FIGURE 2.5: Selected computational grid planes in the primary sub-domain for a representative cantilevered ramp injector.

FIGURE 2.6: Computational grid detail: cross-stream plane just downstream of fuel injection ($\bar{x} = 0$) for a cantilevered ramp injector geometry. Grid mirrored at $y$-axis.
Figure 2.7: Computational grid detail: injector symmetry plane near injector geometry.
This chapter outlines the specifics of the numerical algorithm developed in the current study and subsequently used to solve the various injector flow fields considered. A brief discussion of the physical model employed and the ramifications on the numerical scheme is given first, followed by the governing equations. Details of the spatial and temporal discretization are presented along with boundary conditions and metric relations to complete the numerical method. Validation of the newly developed algorithm is discussed in the final portion of this chapter.

3.1 Physical Modeling Issues

The numerical modeling of physical phenomena to a high degree of fidelity constitutes a considerable challenge. This can be especially true for viscous, high enthalpy, hypervelocity fluid flow simulations which are characterized by high temperatures, shear forces and strong discontinuities. For faithful reproduction of physical phenomena, a numerical model must be chosen which is capable of simulating the subset of physical phenomena present in a given flow field under consideration. Increase in the degree of accuracy of a given numerical model entails a corresponding increase in algorithm complexity. Further, tangible limits exist with regard to computational speed and memory. By judicious selection of the relevant physics to model, an efficient algorithm can be developed which will simulate the phenomena in question to an acceptable degree of accuracy.

It is assumed that the numerical simulations considered can be properly addressed by treat-
ing the gas as a continuum flow. The assumption of negligible rarefied or free molecular effects is valid for flows with Knudsen numbers less than 0.2. The current flow conditions are taken as representative of a shock-induced combustion ramjet flow field where it is expected that the Knudsen number would not exceed $10^{-5}$ at its highest anticipated altitude, based on a practical characteristic vehicle length. Therefore, rarefied flow phenomena are unlikely to be significant and are not considered.

The main focus of the present work is the fuel/air mixing characteristics of various fuel injectors. While the ultimate aim of any fuel injector is efficient mixing which leads to better combustion and ultimately more vehicle thrust, the first stage of assessing the performance of an injector is an understanding of mixing characteristics in the absence of combustion and outside the context of a specific vehicle configuration. Further, fuel injectors applied to shock-induced combustion ramjets are required to mix the fuel and air before combustion. Thus, a chemically non-reacting numerical model is considered.

The mixing mechanisms employed by ramp fuel injectors are largely three dimensional phenomena. Indeed, the main features of interest for enhanced mixing involve axial vortex generation which can not be properly modeled in two dimensions. For this reason, it is fundamental that the present numerical algorithm be formulated in three spatial dimensions.

The focus of the present work is the initial investigation of the new cantilevered ramp injector design with emphasis on the ability of this design to generate large-scale bulk mixing of the fuel and air in the near-field of the injector. This primary mixing is convection dominated and thus, the mixing mechanisms of interest are via gasdynamic processes, such as baroclinic torque vortex generation. Therefore, for this study, turbulence is not modeled and only laminar flow is considered. No effort was made to simulate the turbulence dominated smaller-scale mixing which is most prevalent at large distances downstream of injection.

In high enthalpy flow simulations there are a number of high temperature gas effects to consider. Since combustion is not modeled in the current work and temperatures near the stagnation value are not expected, the maximum temperature encountered is anticipated to be well below the onset of dissociation and ionization. For this reason, no dissociation or ionization effects are considered or modeled. At elevated temperatures, there is an appreciable effect on the specific heat at constant pressure, $C_p$. While there is also a dependence on pressure, (i.e. generally $C_p = C_p(T, P)$) it is manifest only for combinations of very high pressures and low temperatures and thus assumed to be small for the flow fields considered. Thus, $C_p$ is considered to be solely a function of temperature and the gas is said to be thermally perfect.
3.1.1 Overview of Numerical Algorithm

The code used for the present study has been formulated to solve a set of partial differential equations representing the conservation of mass, momentum and energy, more commonly termed the Navier-Stokes equations, for a mixture of laminar, thermally perfect gas species in three generalized coordinates. The largely supersonic flow features require a good shock-capturing scheme which must also perform well when coupled with the high enthalpy nature of the gases under study. A class of upwinding schemes first introduced by Roe is a popular and capable choice for the hypersonic flows considered herein. The present work employs a finite-volume upwind TVD scheme by Yee, combined with the semi-implicit LU-SGS relaxation method developed by Yoon and Jameson.

3.2 Governing Equations

The conservative form of the governing equations in three-dimensional generalized coordinates can be written as

\[
\frac{\partial Q}{\partial t} + \frac{\partial (E - E_v)}{\partial \xi} + \frac{\partial (F - F_v)}{\partial \eta} + \frac{\partial (G - G_v)}{\partial \zeta} = S
\]  

(3.1)

\[
Q = J^{-1} \begin{bmatrix} \rho_1 \\ \vdots \\ \rho_{N_x-1} \\ \rho \\ \rho u \\ \rho v \\ \rho w \\ E \end{bmatrix} \quad ; \quad S = J^{-1} \begin{bmatrix} 0 \\ \vdots \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}
\]  

(3.2)
\[ E = J^{-1} \begin{bmatrix} \rho_1 U \\ \vdots \\ \rho_{N_2-1} U \\ \rho U \\ \rho u U + \xi_s P \\ \rho v U + \xi_v P \\ \rho w U + \xi_w P \\ U (E + P) \end{bmatrix} \quad ; \quad \dot{E}_v = J^{-1} \begin{bmatrix} -\rho_1 \ddot{U}_1 \\ \vdots \\ -\rho_{N_2-1} \ddot{U}_{N_2-1} \\ 0 \\ \xi_x \tau_{xx} + \xi_y \tau_{xy} + \xi_z \tau_{xz} \\ \xi_x \tau_{yx} + \xi_y \tau_{yy} + \xi_z \tau_{yz} \\ \xi_x \tau_{zx} + \xi_y \tau_{zy} + \xi_z \tau_{zz} \\ \xi_x \beta_x + \xi_y \beta_y + \xi_z \beta_z \end{bmatrix} \] (3.3)

\[ F = J^{-1} \begin{bmatrix} \rho_1 V \\ \vdots \\ \rho_{N_1-1} V \\ \rho V \\ \rho u V + \eta_s P \\ \rho v V + \eta_v P \\ \rho w V + \eta_w P \\ V (E + P) \end{bmatrix} \quad ; \quad \dot{F}_v = J^{-1} \begin{bmatrix} -\rho_1 \ddot{V}_1 \\ \vdots \\ -\rho_{N_1-1} \ddot{V}_{N_1-1} \\ 0 \\ \eta_x \tau_{xx} + \eta_y \tau_{xy} + \eta_z \tau_{xz} \\ \eta_x \tau_{yx} + \eta_y \tau_{yy} + \eta_z \tau_{yz} \\ \eta_x \tau_{zx} + \eta_y \tau_{zy} + \eta_z \tau_{zz} \\ \eta_x \beta_x + \eta_y \beta_y + \eta_z \beta_z \end{bmatrix} \] (3.4)

\[ G = J^{-1} \begin{bmatrix} \rho_1 W \\ \vdots \\ \rho_{N_1-1} W \\ \rho W \\ \rho u W + \zeta_s P \\ \rho v W + \zeta_v P \\ \rho w W + \zeta_w P \\ W (E + P) \end{bmatrix} \quad ; \quad \dot{G}_v = J^{-1} \begin{bmatrix} -\rho_1 \ddot{W}_1 \\ \vdots \\ -\rho_{N_1-1} \ddot{W}_{N_1-1} \\ 0 \\ \zeta_x \tau_{xx} + \zeta_y \tau_{xy} + \zeta_z \tau_{xz} \\ \zeta_x \tau_{yx} + \zeta_y \tau_{yy} + \zeta_z \tau_{yz} \\ \zeta_x \tau_{zx} + \zeta_y \tau_{zy} + \zeta_z \tau_{zz} \\ \zeta_x \beta_x + \zeta_y \beta_y + \zeta_z \beta_z \end{bmatrix} \] (3.5)

where the total energy is defined as the sum of internal and kinetic energy as

\[ E = \rho \left[ e + \frac{1}{2} (u^2 + v^2 + w^2) \right] \] (3.6)
and the contravariant velocity components are defined as

\[ U = \xi_x u + \xi_y v + \xi_z w \] (3.7)
\[ V = \eta_x u + \eta_y v + \eta_z w \] (3.8)
\[ W = \zeta_x u + \zeta_y v + \zeta_z w \] (3.9)

The shear stress and heat transfer terms in Eqs. 3.3–3.5 are defined as

\[ \tau_{xx} = 2\mu u_x + \lambda (u_x + v_y + w_z) \] (3.10)
\[ \tau_{xy} = \tau_{yx} = \mu (u_y + v_x) \] (3.11)
\[ \tau_{xz} = \tau_{zx} = \mu (u_z + w_x) \] (3.12)
\[ \tau_{yy} = 2\mu v_y + \lambda (u_x + v_y + w_z) \] (3.13)
\[ \tau_{yz} = \tau_{zy} = \mu (v_z + w_y) \] (3.14)
\[ \tau_{zz} = 2\mu w_z + \lambda (u_x + v_y + w_z) \] (3.15)
\[ \beta_x = u\tau_{xx} + v\tau_{xy} + w\tau_{xz} - q_x \] (3.16)
\[ \beta_y = u\tau_{yx} + v\tau_{yy} + w\tau_{yz} - q_y \] (3.17)
\[ \beta_z = u\tau_{zx} + v\tau_{zy} + w\tau_{zz} - q_z \] (3.18)
\[ q_x = -\kappa \frac{\partial T}{\partial x} + \sum_{k_1=1}^{N_1} h_{k_1}\rho_{k_1}\dot{u}_{k_1} \] (3.19)
\[ q_y = -\kappa \frac{\partial T}{\partial y} + \sum_{k_2=1}^{N_2} h_{k_2}\rho_{k_2}\dot{v}_{k_2} \] (3.20)
\[ q_z = -\kappa \frac{\partial T}{\partial z} + \sum_{k_3=1}^{N_3} h_{k_3}\rho_{k_3}\dot{w}_{k_3} \] (3.21)

where the Cartesian derivatives in the above equations are expanded into generalized coordinates via the chain rule as, for example,

\[ \frac{\partial u}{\partial x} = u_x = \xi_x u_x + \eta_x u_\xi + \zeta_x u_\zeta \] (3.22)

and \( \mu \) is the dynamic viscosity, \( \kappa \) the thermal conductivity and \( \lambda = -\frac{2}{3}\mu \) from the Stokes hypothesis.

The dynamic viscosity of each species is derived from kinetic theory and practically implemented using a polynomial derived using tabulated values from Ref. 61. The viscosity of the
mixture of species is determined using Wilke’s mixing rule\textsuperscript{61}

\[
\mu_{\text{mix}} \equiv \mu = \sum_{k_t=1}^{N_t} \frac{\mu_{k_t}}{1 + \sum_{j_s=1}^{N_t} \frac{X_{j_s}}{X_{k_s}} \phi_{k_s, j_s}}
\]  

(3.23)

where

\[
\phi_{k_s, j_s} = \frac{1}{\sqrt{8}} \left(1 + \frac{M_{k_s}}{M_{j_s}}\right)^{-\frac{1}{2}} \left[1 + \left(\frac{\mu_{k_s}}{\mu_{j_s}}\right)^{\frac{1}{2}} \left(\frac{M_{j_s}}{M_{k_s}}\right)^{\frac{1}{2}}\right]^2
\]  

(3.24)

The thermal conductivity is derived from the kinetic theory of gases and expressed for a pure monoatomic gas as

\[
\kappa_{k_s, o} = \frac{15}{4} \frac{R}{M_{k_s}} \mu_{k_s}
\]  

(3.25)

For a polyatomic gas, the vibrational energy is also considered using the Eucken correction

\[
\kappa_{k_s} = \kappa_{k_s, o} \left(0.115 + 0.354 \frac{C_{p_k}}{R}\right)
\]  

(3.26)

The Mason and Saxena formula is used to calculate the heat transfer coefficient of the mixture as

\[
\kappa_{\text{mix}} \equiv \kappa = \sum_{k_t=1}^{N_t} \frac{\kappa_{k_t}}{1 + \sum_{j_s=1}^{N_t} \frac{X_{j_s}}{X_{k_s}} \phi_{k_s, j_s}}
\]  

(3.27)

where \(\phi_{k_s, j_s}\) is defined in Eq. 3.24.

The diffusion velocities, \(\ddot{u}_{k_s}\), \(\ddot{v}_{k_s}\) and \(\ddot{w}_{k_s}\) are calculated using Fick’s Law as

\[
\ddot{u}_{k_s} = -\frac{D_{k_s, m}}{Y_{k_s}} \frac{\partial Y_{k_s}}{\partial x}
\]  

(3.28)

\[
\ddot{v}_{k_s} = -\frac{D_{k_s, m}}{Y_{k_s}} \frac{\partial Y_{k_s}}{\partial y}
\]  

(3.29)

\[
\ddot{w}_{k_s} = -\frac{D_{k_s, m}}{Y_{k_s}} \frac{\partial Y_{k_s}}{\partial z}
\]  

(3.30)

where \(D_{k_s, m}\) is the multicomponent diffusion coefficient for species \(k_s\) through the mixture. It
can be calculated using the equation

\[ D_{k,m} = \frac{1 - X_{k}}{\sum_{j=1, j \neq k}^{N} \frac{X_{j}}{D_{k,j}}} \]  

(3.31)

where \( D_{k,j} \) is the binary diffusion coefficient between species \( k \) and \( j \) and can be expressed as

\[ D_{k,j} = 5.9543 \times 10^{-24} \sqrt{\frac{T^{2}}{\frac{1}{M_{k}} + \frac{1}{M_{j}}}} \]  

\( \Omega_{k,j}^{(1,1)*} \)  

(3.32)

where, \( \Omega_{k,j}^{(1,1)*} \) is a collision integral and \( \sigma_{k,j} \) is the average collision diameter, both of which are tabulated in Ref. 61.

The contravariant diffusion velocity components are defined for each species \( k \) as

\[ \bar{\rho}_{k} = \bar{\xi}_{x} \bar{u}_{k} + \bar{\xi}_{y} \bar{v}_{k} + \bar{\xi}_{z} \bar{w}_{k} \]  

(3.33)

\[ \bar{v}_{k} = \eta_{x} \bar{u}_{k} + \eta_{y} \bar{v}_{k} + \eta_{z} \bar{w}_{k} \]  

(3.34)

\[ \bar{w}_{k} = \zeta_{x} \bar{u}_{k} + \zeta_{y} \bar{v}_{k} + \zeta_{z} \bar{w}_{k} \]  

(3.35)

The pressure is determined from the equation of state

\[ P = R T \left[ \sum_{k=1}^{N} \left( \frac{\rho_{k}}{M_{k}} \right) \frac{\rho_{k}}{M_{k}} + \frac{\rho}{M_{N}} \right] \]  

(3.36)

Temperature is implicitly determined using an iterative Newton-Raphson procedure from the thermodynamic relation

\[ \sum_{k=1}^{N} \frac{\rho_{k}}{\rho} h_{k} - \sum_{k=1}^{N} \frac{\rho_{k}}{\rho} \frac{RT}{M_{k}} = \frac{E}{\rho} - \frac{1}{2} \left( u^{2} + v^{2} + w^{2} \right) \]  

(3.37)

where the subscript \( k \) represents an individual species. Note that

\[ \rho_{N} = \rho - \sum_{k=1}^{N} \rho_{k} \]

The specific enthalpy of species \( k \) is represented by \( h_{k} \) and includes the heat of formation at
298 K. It is determined from the NASA polynomial curve fits of the JANAF thermochemical tables and is given by

\[ h_{k_s} = \frac{R}{M_{k_s}} T \left( a_{1,k_s} + a_{2,k_s} \frac{T}{2} + a_{3,k_s} \frac{T^2}{3} + a_{4,k_s} \frac{T^3}{4} + a_{5,k_s} \frac{T^4}{5} + \frac{a_{6,k_s}}{T} \right) \]  
(3.38)

where the coefficients \( a_{1,k_s} \ldots a_{6,k_s} \) can be found in Ref. 62. Specific heat at constant pressure, \( C_{p_{k_s}} \), of the species \( k_s \) is given in similar polynomial form by

\[ C_{p_{k_s}} = \frac{R}{M_{k_s}} \left( a_{1,k_s} + a_{2,k_s} T + a_{3,k_s} T^2 + a_{4,k_s} T^3 + a_{5,k_s} T^4 \right) \]  
(3.39)

### 3.3 Temporal Discretization

One way to obtain steady and unsteady solutions to the governing equations is to integrate them in time. This integration can be done explicitly or implicitly.

Discretization of the governing equations can be expressed as

\[ \delta Q = -\beta \Delta t \left[ D_\xi \hat{E}(Q^{n+1}) + D_\eta \hat{F}(Q^{n+1}) + D_\zeta \hat{G}(Q^{n+1}) \right] \]

\[ -(1 - \beta) \Delta t \left[ D_\xi \hat{E}(Q^n) + D_\eta \hat{F}(Q^n) + D_\zeta \hat{G}(Q^n) \right] \]  
(3.40)

where \( \delta Q \) is the correction of the conservative variables used to discretely integrate the solution in time as

\[ Q^{n+1} = Q^n + \delta Q \]  
(3.41)

where the superscripts refer to time step. The flux vectors in Eq. 3.40 are defined as, for example, \( \hat{E} = E - E_v \) and represent the total flux. The terms \( D_\xi, D_\eta, D_\zeta \) are spatial discretization operators used to approximate the partial derivatives: \( \frac{\partial}{\partial \xi}, \frac{\partial}{\partial \eta}, \frac{\partial}{\partial \zeta} \) respectively. The parameter \( \beta \) determines the implicit or explicit nature of the scheme where for \( \beta = 0 \) the scheme is fully explicit while for \( \beta = 1 \) Eq. 3.40 becomes fully implicit. The scheme is second-order accurate in time for a value of \( \beta = \frac{1}{2} \) and first-order time accurate for other values.

From Eq. 3.40 it can be seen that for an implicit scheme the flux vectors must be evaluated at the next \( (n+1) \) time step. In the present formulation only the inviscid flux vectors will be considered implicitly due to the large computational time required for implicit treatment of the viscous fluxes. These terms can be evaluated through the linearization of the flux vectors about
the previous time step as
\[ E(Q^{n+1}) = E(Q^n) + A \delta Q^n + O[(\delta Q)^2] \]  \(3.42\)
\[ F(Q^{n+1}) = F(Q^n) + B \delta Q^n + O[(\delta Q)^2] \]  \(3.43\)
\[ G(Q^{n+1}) = G(Q^n) + C \delta Q^n + O[(\delta Q)^2] \]  \(3.44\)

where the flux Jacobian matrices are defined as
\[ A \equiv \frac{\partial E}{\partial Q}, \quad B \equiv \frac{\partial F}{\partial Q}, \quad C \equiv \frac{\partial G}{\partial Q} \]  \(3.45\)

Substitution of Eqs. 3.42 to 3.44 into Eq. 3.40 and neglecting terms of second and higher order, results in
\[ \left[ I + \beta \Delta t (D_\xi A + D_\eta B + D_\zeta C) \right] \delta Q = -\Delta t \mathbf{R} \]  \(3.46\)

where the residual, \( \mathbf{R} \), is defined by
\[ \mathbf{R} = D_\xi (E - E_v) + D_\eta (F - F_v) + D_\zeta (G - G_v) \]  \(3.47\)

Inspection of Eq. 3.46 reveals that direct inversion of the factor premultiplying \( \delta Q \) (left hand side term) becomes impractical in three dimensions due to its large size. The computational cost for direct inversion of large matrices also increases non-linearly with increasing matrix elements which makes this method unacceptable for problems with many computational cells. In order to alleviate this difficulty, much research has focused on a way to factorize the implicit (LHS) operator, usually in an approximate manner.\(^{63-69}\) The method of factorization must be chosen carefully to insure stable temporal discretization.

Yoon and Jameson\(^{60}\) derived an implicit algorithm by combining LU factorization and SGS relaxation. This method is stable in any number of spatial dimensions and has the benefit of accelerated convergence with computational costs on the order of explicit methods. The so-called, LU-SGS scheme can be written as a product of three factors as
\[ L D^{-1} U \delta Q = -\Delta t \mathbf{R} \]  \(3.48\)
where
\[
L = I + \beta \Delta t \left( D_\zeta \mathbf{A}^+ + D_\eta \mathbf{B}^+ + D_\zeta \mathbf{C}^+ - \mathbf{A}^- - \mathbf{B}^- - \mathbf{C}^- \right) \\
D = I + \beta \Delta t \left( \mathbf{A}^+ - \mathbf{A}^- + \mathbf{B}^+ - \mathbf{B}^- + \mathbf{C}^+ - \mathbf{C}^- \right) \\
U = I + \beta \Delta t \left( D_\zeta \mathbf{A}^- + D_\eta \mathbf{B}^- + D_\zeta \mathbf{C}^- + \mathbf{A}^+ + \mathbf{B}^+ + \mathbf{C}^+ \right)
\] (3.49) (3.50) (3.51)

The flux Jacobian matrices are split such that discretization can be carried out in an upwind fashion based on characteristic directions. In order to insure diagonal dominance of the flux Jacobian matrices, and hence stable convergence, they are constructed in such a manner so that "+" matrices have non-negative eigenvalues, while "−" matrices have non-positive eigenvalues. One choice for this construction is
\[
\mathbf{A}^\pm = \frac{1}{2} (\mathbf{A} \pm \nu_A I) \\
\mathbf{B}^\pm = \frac{1}{2} (\mathbf{B} \pm \nu_B I) \\
\mathbf{C}^\pm = \frac{1}{2} (\mathbf{C} \pm \nu_C I)
\] (3.52) (3.53) (3.54)

where, for example
\[
\nu_A = \chi \max \{ |\lambda(\mathbf{A})| \}
\] (3.55)

where \( \lambda(\mathbf{A}) \) represents the eigenvalues of the flux Jacobian \( \mathbf{A} \) and \( \chi \) is a constant \( \geq 1 \).

The LU-SGS algorithm can be inverted in three steps as
\[
\delta \mathbf{Q}^* = -\Delta t \mathbf{L}^{-1} \mathbf{R} \\
\delta \mathbf{Q}^{**} = \mathbf{D} \delta \mathbf{Q}^* \\
\delta \mathbf{Q} = \mathbf{U}^{-1} \delta \mathbf{Q}^{**}
\] (3.56) (3.57) (3.58)

If we set \( \beta = 1 \) and \( \Delta t \to \infty \) we obtain the so-called Newton-like iteration form. Because of approximate factorization, the term Newton iteration is not strictly accurate. In this form, the advantages of rapid convergence toward steady-state and the elimination of arbitrary parameters, such as optimal choice of Courant number, are realized. If we further choose the
discretization operators of the flux Jacobians to be

\[
\begin{align*}
D_\xi^- A^+ &= A_{i,j,k}^+ - A_{i-1,j,k}^+ & \quad (3.59) \\
D_\xi^+ A^- &= A_{i+1,j,k}^- - A_{i,j,k}^- & \quad (3.60) \\
D_\eta^- B^+ &= B_{i,j,k}^- - B_{i,j-1,k}^+ & \quad (3.61) \\
D_\eta^+ B^- &= B_{i,j+1,k}^- - B_{i,j,k}^- & \quad (3.62) \\
D_\zeta^- C^+ &= C_{i,j,k}^+ - C_{i,j,k-1}^+ & \quad (3.63) \\
D_\zeta^+ C^- &= C_{i,j,k+1}^- - C_{i,j,k}^- & \quad (3.64)
\end{align*}
\]

Eqs. 3.49 to 3.51 reduce to

\[
\begin{align*}
L &= \tilde{\nu} I - A_{i-1,j,k}^+ - B_{i,j-1,k}^+ - C_{i,j,k-1}^+ \\
D &= \tilde{\nu} I \\
U &= \tilde{\nu} I + A_{i+1,j,k}^- + B_{i,j+1,k}^- + C_{i,j,k+1}^-
\end{align*}
\]

where

\[
\tilde{\nu} = \nu_A + \nu_B + \nu_C \quad (3.68)
\]

This form of the algorithm provides for an implicit method without the need for costly block-matrix inversions. Indeed the algorithm requires only scalar inversions for each of the factors in Eqs. 3.65 to 3.67 when calculations are carried out sequentially along \( i + j + k = \text{const.} \) oblique planes of sweep. This can be seen by inspection of a schematic representation of

**Figure 3.1:** Schematic representation of LU-SGS algorithm.
Eq. 3.48 (employing the factors given by Eqs. 3.65 to 3.67) given in Fig. 3.1. In this figure the squares in each large matrix represent a "block" of entries of dimension $n \times n$ and the rectangles represent vectors of length $n$, where $n$ is the number of governing equations. By back substitution of these factors in the indicated "sweep direction" it can be seen that direct inversion of the large matrix factors can be avoided. Note also that the shaded diagonal entries in this case involve only diagonal elements within the block allowing scalar inversion.

3.4 Spatial Discretization: Shock-Capturing Scheme

The residual in Eqn. 3.46 can be written in two components as

$$ R = R_I - R_V $$  \hspace{1cm} (3.69)

where

$$ R_I = D_{\xi}E + D_{\eta}F + D_{\zeta}G $$  \hspace{1cm} (3.70)

$$ R_V = D_{\xi}E_v + D_{\eta}F_v + D_{\zeta}G_v $$  \hspace{1cm} (3.71)

represent the inviscid and viscous portions of the residual respectively. Note that the discretization operators acting on the inviscid and viscous components of the residual are not required to be the same.

The viscous portion of the residual (Eq. 3.71) was discretized using cell-centered second-order central difference stencils.

In order to calculate the inviscid residual in Eqn. 3.69, a total variation diminishing (TVD) scheme proposed by Yee was used. A class of TVD schemes has been defined which can be grouped into two main categories as symmetric and upwind formulations. For upwind schemes the numerical dissipation depends on the signs of the characteristic speeds while for the symmetric class the dissipation is independent of the direction of the characteristic speeds. All schemes presented herein are second-order spatially accurate in smooth regions of the flow.

The inviscid residual can be expressed as a function of numerical flux terms as

$$ R_I = - \left[ \tilde{E}_{i+\frac{1}{2},j,k} - \tilde{E}_{i-\frac{1}{2},j,k} + \tilde{F}_{i,j+\frac{1}{2},k} - \tilde{F}_{i,j-\frac{1}{2},k} + \tilde{G}_{i,j,k+\frac{1}{2}} - \tilde{G}_{i,j,k-\frac{1}{2}} \right] $$  \hspace{1cm} (3.72)
where, for example, the numerical flux function $\tilde{E}_{i+\frac{1}{2},j,k}$ is written as
\begin{equation}
\tilde{E}_{i+\frac{1}{2},j,k} = \frac{1}{2} \left[ E_{i+1,j,k} + E_{i,j,k} + (R_\xi \Phi)_{i+\frac{1}{2},j,k} / J_{i+\frac{1}{2},j,k} \right]
\end{equation}

where $R_\xi$ is defined as the matrix whose columns are the right eigenvectors of the inviscid flux Jacobian $A$, and the metric Jacobian at the cell boundary is taken as the arithmetic average of the adjacent cell centre values as
\begin{equation}
\frac{1}{J_{i+\frac{1}{2},j,k}} = \frac{1}{2} \left( \frac{1}{J_{i+1,j,k}} + \frac{1}{J_{i,j,k}} \right)
\end{equation}

It is important to note that if the metric terms (such as $\xi_x/J$) in the inviscid flux vectors of Eqn. 3.73 are evaluated at the cell interface, i.e. at $()_{i+\frac{1}{2},j,k}$, the resulting numerical flux function is in a true finite volume formulation. This is to be compared with the case where the flow variables and the metric terms appearing in the inviscid flux vectors are both taken to be the respective cell-centered values; in this case the numerical flux vector is considered to be in a pseudo finite volume form. The former formulation is recommended for highly skewed grids and nonuniform flows since it preserves freestream flow in all cases (see Ref. 70 for a discussion). Thus, this formulation is adopted in the present work.

Equation 3.73 is common to both the symmetric and upwind schemes. The form of the vector $\Phi_{i+\frac{1}{2},j,k}$ differs in the two formulations however. Specifically, for the symmetric scheme, the elements of $\Phi_{i+\frac{1}{2},j,k}$ in the $\xi$-direction denoted by $(\phi'_{i+\frac{1}{2},j,k})^S$ for a spatially second-order TVD formulation can be expressed as
\begin{equation}
(\phi'_{i+\frac{1}{2},j,k})^S = -\psi \left( \lambda^l_{i+\frac{1}{2},j,k} \right) \left[ \alpha^l_{i+\frac{1}{2},j,k} - g^l_{i+\frac{1}{2},j,k} \right]
\end{equation}

where $\lambda^l_{i+\frac{1}{2},j,k}$ represents the eigenvalue (characteristic speed) of $A$ evaluated at a symmetric average (such as Roe averaging) between the states $Q_{i,j,k}$ and $Q_{i+1,j,k}$. The values $\alpha^l_{i+\frac{1}{2},j,k}$ are elements of the vector
\begin{equation}
\alpha_{i+\frac{1}{2},j,k} = R_\xi^{-1}_{i+\frac{1}{2},j,k} \left( J_{i+1,j,k} Q_{i+1,j,k} - J_{i,j,k} Q_{i,j,k} \right)
\end{equation}

and represent jumps in the the characteristic variables. At first blush, Eqn. 3.76 looks to be computationally expensive but one can take advantage of the form of the matrices involved to derive a very computationally efficient form (see Sec. 3.4.1). The variable $g^l_{i+\frac{1}{2},j,k}$ in Eqn. 3.75 represents elements of the so-called "limiter" function which affords the TVD scheme second
order accuracy. Examples of these functions can be expressed as:

\[
g_{i+\frac{1}{2},j,k} = \min \left( \alpha_{i-\frac{1}{2},j,k}, \alpha_{i+\frac{1}{2},j,k} \right) + \min \left( \alpha_{i+\frac{1}{2},j,k}, \alpha_{i+\frac{3}{2},j,k} \right) - \alpha_{i+\frac{1}{2},j,k} \\
g_{i+\frac{3}{2},j,k} = \min \left( \alpha_{i-\frac{1}{2},j,k}, \alpha_{i+\frac{1}{2},j,k}, \alpha_{i+\frac{3}{2},j,k} \right) \\
g_{i+\frac{3}{2},j,k} = \min \left[ 2 \alpha_{i-\frac{1}{2},j,k}, 2 \alpha_{i+\frac{1}{2},j,k}, 2 \alpha_{i+\frac{3}{2},j,k}, \frac{1}{2} \left( \alpha_{i-\frac{1}{2},j,k} + \alpha_{i+\frac{1}{2},j,k} \right) \right]
\]

(3.77) (3.78) (3.79)

The minmod function returns the argument with the smallest modulus from a list of arguments when they are of the same sign and zero otherwise. This can be written in compact form suitable for practical implementation as

\[
\text{minmod}(x,y) = \text{sgn}(x) \cdot \max[0, \min(|x|, \text{sgn}(x) \cdot y)]
\]

(3.80)

\[
\text{minmod}(x,y,z) = \text{sgn}(x) \cdot \max[0, \min(|x|, \text{sgn}(x) \cdot y, \text{sgn}(x) \cdot z)]
\]

(3.81)

The limiter functions in Eqns 3.77 - 3.79 are listed in order of decreasing diffusivity and thus sharper resolution of discontinuities. This increase in accuracy however comes at a cost of decreased stability and convergence rate.

In the case of the upwind scheme, the elements of \( \Phi_{i+\frac{1}{2},j,k} \) in the \( \xi \)-direction denoted by \( (\phi'_{i+\frac{1}{2},j,k})^U \) for a spatially second-order TVD formulation can be expressed as

\[
(\phi'_{i+\frac{1}{2},j,k})^U = \frac{1}{2} \psi \left( \lambda'_{i+\frac{1}{2},j,k} \right) (g_{i+1,j,k} + g_{i,j,k}) - \psi \left( \lambda'_{i+\frac{1}{2},j,k} + \gamma'_{i+\frac{1}{2},j,k} \right) \alpha'_{i+\frac{1}{2},j,k}
\]

(3.82)

where,

\[
\gamma'_{i+\frac{1}{2},j,k} = \frac{1}{2} \psi \left( \lambda'_{i+\frac{1}{2},j,k} \right) \left\{ \begin{array}{ll} \\
\frac{g_{i+1,j,k} - g_{i,j,k}}{\alpha'_{i+\frac{1}{2},j,k}} & \alpha'_{i+\frac{1}{2},j,k} \neq 0 \\
0 & \alpha'_{i+\frac{1}{2},j,k} = 0
\end{array} \right.
\]

(3.83)
Limiter functions suitable for the upwind formulation include\textsuperscript{71}

\begin{align}
g_{i,j,k}^l & = \text{minmod} \left( \alpha_{i-\frac{1}{2},j,k}^l, \alpha_{i+\frac{1}{2},j,k}^l \right) \\
g_{i,j,k} & = \frac{\left( \alpha_{i-\frac{1}{2},j,k}^l, \alpha_{i+\frac{1}{2},j,k}^l \right) + \left( \alpha_{i+\frac{1}{2},j,k}^l, \alpha_{i+\frac{1}{2},j,k}^l \right) + \delta}{\left( \alpha_{i+\frac{1}{2},j,k}^l \right)^2 + \delta} \\
g_{i,j,k} & = \text{minmod} \left( 2 \alpha_{i-\frac{1}{2},j,k}^l, 2 \alpha_{i+\frac{1}{2},j,k}^l, \frac{1}{2} \left[ \alpha_{i+\frac{1}{2},j,k}^l, \alpha_{i+\frac{1}{2},j,k}^l \right] \right) \\
g_{i,j,k} & = S \cdot \max \left[ 0, \min \left( 2 \left| \alpha_{i+\frac{1}{2},j,k}^l \right|, S \cdot \alpha_{i+\frac{1}{2},j,k}^l \right) \right]
\end{align}

where $\delta$ is a small parameter on the order of $10^{-15}$ to prevent division by zero and $S = \text{sgn}(\alpha_{i+\frac{1}{2},j,k}^l)$.

Equations 3.84 to 3.88 are listed in order of decreasing difusiveness, and thus ease of convergence, and increasing accuracy. It has been suggested by Yee that limiters 3.87 and 3.88 can be unstable and may not suitable for all problems.\textsuperscript{71}

In both the symmetric and upwind formulations the function $\psi$ is an entropy satisfying function used to omit aphysical solutions. It can be generally expressed as

\[ \psi(\lambda) = \begin{cases} 
|\lambda|, & |\lambda| \geq \varepsilon \\
(\lambda^2 + \varepsilon^2)/2\varepsilon, & |\lambda| < \varepsilon 
\end{cases} \]

where the eigenvalue limiter $\varepsilon$ is a small positive parameter. Yee\textsuperscript{71} asserts that for steady state computations, particularly hypersonic blunt body flows, $\varepsilon$ should be a function of velocity and sound speed to prevent instabilities, in particular, reversed flow in the stagnation region, leading to the following formulation

\[ \varepsilon_{i+\frac{1}{2},j,k} = \bar{\varepsilon} \left[ |U_{i+\frac{1}{2},j,k}| + |V_{i+\frac{1}{2},j,k}| + |W_{i+\frac{1}{2},j,k}| + \frac{a_{i+\frac{1}{2},j,k}}{3} \left( k_{i+\frac{1}{2},j,k}^t + k_{i+\frac{1}{2},j,k}^w + k_{i+\frac{1}{2},j,k}^c \right) \right] \]

where $k_{i+\frac{1}{2},j,k}^t$ is defined as

\[ k_{i+\frac{1}{2},j,k}^t = \sqrt{\frac{\varepsilon_t^2}{\xi_t^2}} \]

Moreover, the parameter $\varepsilon_{i+\frac{1}{2},j,k}$ does not only serve to exclude non-physical solutions but
also has an effect on the convergence rate and shock resolution of the flow solution. For very small values of this variable, discontinuities are sharper but the convergence rate is slower and perhaps less stable.\textsuperscript{71}

### 3.4.1 Efficient Computational Forms

It has been proposed that the vector $\alpha_{i+\frac{1}{2},j,k}$ and the matrix product $(R_\Phi)_{i+\frac{1}{2},j,k}$ in Eqs. 3.76 and 3.73 respectively be recast into a form more computationally efficient for practical implementation.\textsuperscript{59} Defining the operator

$$\Delta(\cdot)' = (\cdot)'_{i+1,j,k} - (\cdot)'_{i,j,k}$$

Eqn. 3.76 can be written explicitly as

$$\alpha_{i+\frac{1}{2},j,k} = \left[ \begin{array}{c} \Delta P_1 - \frac{\delta x}{\delta^2} \delta P \\ \vdots \\ \Delta P_{N_t-1} - \frac{\delta x_{N_t-1}}{\delta^2} \delta P \\ \Delta P - \sum_{k=1}^{N_t} \Delta P_k - \frac{\delta x_k}{\delta^2} \delta P \\ -\bar{U}_s \Delta P + \bar{s}_1 \Delta (\rho u) + \bar{s}_2 \Delta (\rho v) + \bar{s}_3 \Delta (\rho w) \\ -\bar{U}_f \Delta P + \bar{f}_1 \Delta (\rho u) + \bar{f}_2 \Delta (\rho v) + \bar{f}_3 \Delta (\rho w) \\ \frac{1}{2\delta^2} [\delta P - \bar{a} (\Delta P \bar{U}_k - \mathcal{B})] \\ \frac{1}{2\delta^2} [\delta P + \bar{a} (\Delta P \bar{U}_k - \mathcal{B})] \end{array} \right]$$

(3.92)

where

$$\mathcal{B} = \bar{k}_1 \Delta (\rho u) + \bar{k}_2 \Delta (\rho v) + \bar{k}_3 \Delta (\rho w)$$

and

$$\delta P = \sum_{k=1}^{N_t-1} \Delta P_k \bar{P}_{k_s} + \Delta P \bar{P}_p - \bar{P}_E (\bar{u} \Delta (\rho u) + \bar{v} \Delta (\rho v) + \bar{w} \Delta (\rho w) - \Delta E)$$

(3.93)

The term $\delta P$ is the differential of pressure on a discrete mesh. This, it has been proposed\textsuperscript{72} that Eqn. 3.93 be replaced with

$$\delta P = P_{i+1,j} - P_{i,j}$$

(3.94)
which is slightly less diffusive and more computationally efficient with negligible affect on solution accuracy.

The quantities with an overbar represent values averaged at some symmetric average between \( i \) and \( i + 1 \). This averaging can be a simple arithmetic average but more often Roe averaging is used for increased accuracy. A summary of Roe averaging for a multi-species gas can be found in Liu et al.\(^{73}\)

The matrix product \((R_r \Phi)_{i,i+\frac{1}{2},j,k}\) can be written in a convenient form which takes advantage of the special structure of the component matrices. With straightforward algebraic manipulations the matrix product becomes

\[
(R_r \Phi)_{i,i+\frac{1}{2},j,k} = \begin{bmatrix}
\phi_1 + \tilde{y}_1 \mathcal{K}_1 \\
\vdots \\
\phi_{N_r-1} + \tilde{y}_{N_r-1} \mathcal{K}_1 \\
\mathcal{K}_2 \\
\tilde{u} \mathcal{K}_2 + \tilde{k}_1 \mathcal{K}_3 + \tilde{s}_1 \phi_{N_r+1} + \tilde{t}_1 \phi_{N_r+2} \\
\tilde{v} \mathcal{K}_2 + \tilde{k}_2 \mathcal{K}_3 + \tilde{s}_2 \phi_{N_r+1} + \tilde{t}_2 \phi_{N_r+2} \\
\tilde{w} \mathcal{K}_2 + \tilde{k}_3 \mathcal{K}_3 + \tilde{s}_3 \phi_{N_r+1} + \tilde{t}_3 \phi_{N_r+2} \\
\sum_{k_r=1}^{N_r-1} \phi_{k_r} \left[ \mathcal{K}_4 - \frac{\tilde{p}_{k_r}}{\tilde{p}_E} \right] + \mathcal{K}_4 \phi_{N_r} + \tilde{u}_s \phi_{N_r+1} + \tilde{u}_t \phi_{N_r+2} + \mathcal{K}_3 \tilde{U}_k + \mathcal{K}_1 \tilde{h}_0
\end{bmatrix}
\]

where

\[
\mathcal{K}_1 = \phi_{N_r} + \phi_N \tag{3.96}
\]

\[
\mathcal{K}_2 = \sum_{k_r=1}^{N_r} \phi_{k_r} + \mathcal{K}_1 \tag{3.97}
\]

\[
\mathcal{K}_3 = \tilde{a} (\phi_{N_r+1} + \phi_N) \tag{3.98}
\]

\[
\mathcal{K}_4 = \tilde{q}_s^2 - \frac{\tilde{p}_s}{\tilde{p}_E} \tag{3.99}
\]

with similar expressions for \((R_l \Phi)_{i,i+\frac{1}{2},j} \) and \((R_c \Phi)_{i,j,i+\frac{1}{2}} \).
For notational convenience the following terms are also defined

\[
\begin{align*}
  k_1 &= \frac{\xi_x}{\xi_1} / k_{\xi} \\
  k_2 &= \frac{\xi_y}{\xi_2} / k_{\xi} \\
  k_3 &= \frac{\xi_z}{\xi_3} / k_{\xi} \\
  t_1 &= \frac{k_3 - k_2}{d} \\
  t_2 &= \frac{k_1 - k_3}{d} \\
  t_3 &= \frac{k_2 - k_1}{d} \\
  s_1 &= k_2 t_3 - k_3 t_2 \\
  s_2 &= k_3 t_1 - k_1 t_3 \\
  s_3 &= k_1 t_2 - k_2 t_1
\end{align*}
\] (3.100)

\[
  k_{\xi} = \sqrt{\left(\frac{\xi_x}{\xi_1}\right)^2 + \left(\frac{\xi_y}{\xi_2}\right)^2 + \left(\frac{\xi_z}{\xi_3}\right)^2}
\] (3.103)

\[
d^2 = (k_3 - k_2)^2 + (k_1 - k_3)^2 + (k_2 - k_1)^2
\] (3.104)

\[
U_k = k_1 u + k_2 v + k_3 w
\] (3.105)

\[
U_t = t_1 u + t_2 v + t_3 w
\] (3.106)

\[
U_s = s_1 u + s_2 v + s_3 w
\] (3.107)

3.5 Boundary Conditions

The numerical implementation of the physical boundary conditions must be handled with care in order to insure fidelity and stability. Indeed, the driving force of the flow field solution is the conditions imposed at the domain boundaries which underscores the sensitivity of their treatment to the final solution. Further, numerical implementation must be consistent with the numerical scheme employed to ensure convergence and stability.

A number of different boundary conditions are considered for the present numerical scheme. Transmissive boundaries, where flow enters or exist the domain, solid surface, or wall boundaries, and the symmetric boundary condition are outlined in the following sections.

3.5.1 Inflow/Outflow Conditions

The specification of flow variables at inflow and outflow domain boundaries should be consistent with the physics of the flow at that boundary. Specifically, the physical information entering or exiting the domain should dictate boundary conditions. This information is represented by the characteristic directions of the flow. For supersonic flow all characteristics are of
the same sign (i.e. traveling in the same direction). Thus, for locally supersonic flow, all information enters at inflow boundaries and all information leaves the domain at outflow boundaries. Practically, this leads to the specification of all flow variables at supersonic inflow boundaries and the (zero-order) extrapolation of all flow variables at supersonic outflow boundaries.

### 3.5.2 Surface Boundary Condition

The solution of the Navier-Stokes and Euler equations dictate different physical boundary conditions to be imposed at domain boundaries bordering solid surfaces. In the case of the latter, the so called "slip" condition is imposed on the velocity at the wall. This condition simply states that the flow velocity at a wall boundary must be tangent to said wall. This dictates that the velocity normal to the solid wall must be zero. It can be expressed mathematically as

\[
(V \cdot n)_{wall} = 0
\]

(3.108)

where \( V \) is the velocity vector and \( n \) is the unit vector normal to the wall. In the context of the present method employing generalized coordinates we have

\[
\begin{bmatrix}
\nu \\
\nu \\
w
\end{bmatrix}
= J^{-1}
\begin{bmatrix}
(\eta_\xi \zeta_x - \eta_x \zeta_\xi) & -(\xi_\eta \zeta_x - \xi_x \zeta_\eta) & (\xi \eta_x \zeta - \eta_x \xi_x) \\
-(\eta_\xi \zeta_x - \eta_x \zeta_\xi) & (\xi_\eta \zeta_x - \xi_x \zeta_\eta) & -(\xi \eta_x \zeta - \eta_x \xi_x) \\
(\eta_\xi \zeta_x - \eta_x \zeta_\xi) & -(\xi_\eta \zeta_x - \xi_x \zeta_\eta) & (\xi \eta_x \zeta - \eta_x \xi_x)
\end{bmatrix}
\begin{bmatrix}
U \\
V \\
W
\end{bmatrix}
\]

(3.109)

where the non-tangential velocity component is set equal to zero at a given wall (for example, \( W = 0 \) for a solid boundary along the surface \( \zeta = \text{const.} \)). This is the only physical condition to be implemented at a solid surface for the Euler equations. The remaining flow variables at the wall are extrapolated from the interior flow field solution. This extrapolation of the variables to the cell surface comprising a wall boundary condition can be done in a linear fashion or such that the gradients of the variables are zero (zero order extrapolation).

For the treatment of a wall boundary condition in the solution of the Navier-Stokes implementation, conditions are dictated by the physics of viscous flow. The first of these conditions is imposed on the velocity at the wall. In a viscous flow the velocity at a solid surface must be zero. This so-called, "no-slip" condition, for a stationary wall, may be stated as

\[
V|_{wall} = 0
\]

(3.110)
or explicitly

\[ u = 0 \quad ; \quad v = 0 \quad ; \quad w = 0 \]  \hspace{1cm} (3.111)

Secondly, a condition on the wall temperature must be imposed. This can take the form of specifying a constant wall temperature or constant wall heat flux. Mathematically we have

\[ T_{\text{wall}} = T_w = \text{constant} \]  \hspace{1cm} (3.112)

or

\[ -\kappa \frac{\partial T}{\partial n}_{\text{wall}} = \dot{q}_w = \text{constant} \]  \hspace{1cm} (3.113)

where \( \dot{q}_w \) is the specified wall heat flux. For the case of an adiabatic wall \( q_w = 0 \).

In order to attain thermodynamic closure at a solid surface an additional flow variable must be specified. It has been proposed that that the normal pressure gradient at the wall be taken to be zero: \(^{74,75}\)

\[ \frac{\partial p}{\partial n}_{\text{wall}} = 0 \]  \hspace{1cm} (3.114)

This is implemented in the present solver by setting the wall pressure equal to the pressure at the adjacent cell centre.

### 3.5.3 Symmetry Condition

At domain boundaries which represent a symmetry plane of the physical domain the following conditions are specified. The velocity vector at a symmetry boundary is specified as tangent to the boundary using Eq. 3.109. All the remaining flow variables are specified using the condition of vanishing normal gradient at the wall

\[ \frac{\partial Q}{\partial n}_{\text{wall}} = 0 \]  \hspace{1cm} (3.115)

### 3.6 Curvilinear Transformations: Metrics

The governing equations given in Sec. 3.2 are expressed in generalized curvilinear coordinates. This formulation transforms the equations from non-uniform Cartesian physical space to a uni-
formly spaced computational domain. This gives the flexibility of specifying a structured grid with arbitrary spacing and clustering in physical space while treating the equations consistently in the computational algorithm via the transformation to the computational domain. In general, the generalized coordinates are a function of the Cartesian coordinates so that

\[
\begin{align*}
\xi &= \xi(x, y, z) \\
\eta &= \eta(x, y, z) \\
\zeta &= \zeta(x, y, z)
\end{align*}
\]

This transformation from the physical to the computational domain is afforded by the metric terms in the governing equations (e.g., $\xi_x$, $\xi_y$) which describe the relationship between the coordinate systems. Since it is more computationally convenient to compute the inverse of these terms, i.e., $x_\xi, y_\xi$, the following relations, obtained from simple chain-rule expansions, are employed

\[
\begin{align*}
\xi_x &= J(y_\eta z_\zeta - y_\zeta z_\eta) \\
\xi_y &= J(z_\eta x_\zeta - x_\eta z_\zeta) \\
\xi_z &= J(x_\eta y_\zeta - y_\eta x_\zeta) \\
\eta_x &= J(z_\xi y_\zeta - y_\xi z_\zeta) \\
\eta_y &= J(x_\xi z_\zeta - x_\xi y_\zeta) \\
\eta_z &= J(y_\xi x_\zeta - x_\xi y_\zeta) \\
\zeta_x &= J(y_\xi z_\eta - z_\xi y_\eta) \\
\zeta_y &= J(x_\xi z_\eta - x_\xi y_\eta) \\
\zeta_z &= J(x_\xi y_\eta - y_\xi x_\eta)
\end{align*}
\]

where $J$ is termed the metric Jacobian and is given by

\[
J^{-1} = x_\xi y_\eta z_\zeta + x_\zeta y_\xi z_\eta + x_\eta y_\zeta z_\xi - x_\eta y_\zeta z_\xi - x_\xi y_\xi z_\eta - x_\xi y_\eta z_\zeta
\]

where all subscripts in the above relations denote partial differentiation.

### 3.6.1 Metric Discretization

The spatial discretization of the metric terms deserves due diligence to preserve the overall fidelity of the numerical scheme. In practice it is most common to evaluate the metric deriva-
tives using centered second-order finite difference stencils. Within this constraint there remain several choices as to how this is implemented. The expressions employed can make use of the values of the cell centers, cell vertices or a combination of both. Further, treatment of the metric terms at computational boundaries must be handled with care. For a numerical scheme employing a finite difference formulation a significant error can be introduced if the metric terms are discretized differently from the governing equations. This will manifest itself in the inability of a given scheme to maintain uniform freestream flow. In the present scheme a finite volume formulation is employed which guarantees the maintenance of freestream flow and thus reduces any error introduced by the metric terms. Indeed, uniform flow is an exact solution of the governing equations discretized in a finite volume manner. Thus, in the present scheme, the metric derivatives have been calculated using second-order finite difference stencils employing the cell vertices, with special care taken to preserve second-order accuracy at domain boundaries.

The metric terms at the cell boundaries in the \( \xi \)-direction can be evaluated as

\[
x_{\xi} \big|_{i-\frac{1}{2}} = \frac{1}{2} \left( \frac{x_{i-\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - x_{i-\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + x_{i-\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - x_{i-\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \eta} \right)
\]

\[
y_{\xi} \big|_{i-\frac{1}{2}} = \frac{1}{2} \left( \frac{y_{i-\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - y_{i-\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + y_{i-\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - y_{i-\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \eta} \right)
\]

\[
z_{\xi} \big|_{i-\frac{1}{2}} = \frac{1}{2} \left( \frac{z_{i-\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - z_{i-\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + z_{i-\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - z_{i-\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \eta} \right)
\]

\[
x_{\xi} \big|_{i+\frac{1}{2}} = \frac{1}{2} \left( \frac{x_{i+\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - x_{i+\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + x_{i+\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - x_{i+\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \zeta} \right)
\]

\[
y_{\xi} \big|_{i+\frac{1}{2}} = \frac{1}{2} \left( \frac{y_{i+\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - y_{i+\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + y_{i+\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - y_{i+\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \zeta} \right)
\]

\[
z_{\xi} \big|_{i+\frac{1}{2}} = \frac{1}{2} \left( \frac{z_{i+\frac{1}{2}, j+\frac{1}{2}, k+\frac{1}{2}} - z_{i+\frac{1}{2}, j-\frac{1}{2}, k+\frac{1}{2}} + z_{i+\frac{1}{2}, j+\frac{1}{2}, k-\frac{1}{2}} - z_{i+\frac{1}{2}, j-\frac{1}{2}, k-\frac{1}{2}}}{\Delta \zeta} \right)
\]

with similar expression for the derivatives in the \( \eta \) and \( \zeta \) directions.

The metric terms at the cell centers are evaluated using an average of the cell face values, for example,

\[
\left. \frac{\xi_{\xi}}{J} \right|_{i} = \frac{1}{2} \left[ \left. \frac{\xi_{\xi}}{J} \right|_{i-\frac{1}{2}} + \left. \frac{\xi_{\xi}}{J} \right|_{i+\frac{1}{2}} \right]
\]

with similar expression for the other metric terms.
3.7 Code Validation

The complete validation of a new flowsolver has several components. The first of these components should be rudimentary internal consistency checks. These checks would include the ability of the code to maintain a uniform freestream flow and the consistent treatment of boundary conditions for arbitrarily oriented flow (i.e. flow in the positive direction should yield a solution that is mirror to the negative direction). Other simple checks may include observations that the code gives intuitive physical solutions. Comparison with simple theory, where applicable, can be done on a limited basis such as for shock angles and property ratios across discontinuities.

The second stage in validation can consist of comparison of simple flow fields with existing, and known to be “working”, flowsolvers. This can be broken down into several components also such that comparisons can be made for Euler and Navier-Stokes solutions followed by two and three-dimensional cases. Progressively more complicated flow fields can be solved and compared until the user is satisfied that the new code gives very similar results to the established codes for several cases.

The third main component in a proper code validation is comparison of the final algorithm with experimentally measured results. It can be said that this is the only “true” validation of a new code and is a necessary step in asserting that the code models real physics with an acceptable degree of accuracy. Indeed, this degree of accuracy is paramount in determining the confidence level of subsequent solutions. It is clear that comparison with experimental data can be done for only a limited number of cases and does not ensure that the numerical algorithm will perform properly in all subsequent cases, but a proper and thorough validation process should lend the code adequate credibility. The limited number of experimental test cases is particularly acute in the case of hypersonic flow. It is hoped that the final stage of validation will test and verify proper operation of all parts of the flowsolver since favourable comparison with experimental data is only possible when all aspects of the code function properly and accurately.

The present code has been rigorously tested for internal consistency checks and coding errors. Comparison with simple flow fields where analytical solutions are available have been investigated with excellent results. Comparison with existing flow solvers using the Euler formulation of the present code has also been completed for multiple flow fields with good agreement.

Several validation cases have been solved in order to address the aforementioned validation process in the context of the full Navier-Stokes solver formulation. The following sections give details of these cases in an effort to demonstrate the accuracy of the present code and its
acceptability in solving the problems within the scope of the present work.

3.7.1 Flat Plate and Plane

As a first attempt to ascertain the proper operation of the code in solving the Navier-Stokes equations, a flat plate case was solved in every possible direction in two and three dimensions (i.e., flow in the positive $\xi$, negative $\xi$, positive $\eta$, etc.). Further, the wall boundary condition was implemented on all possible faces to test the implementation of the Navier-Stokes conditions at a solid surface. This test consisted of 12 cases in the three dimensional case and 8 cases for the two-dimensional case. Care was taken to ensure the proper and consistent grid clustering at the viscous wall boundary for all cases. The convergence histories and the final solutions of each case was compared for consistency. The three-dimensional cases were flat plane analogies of the two-dimensional flat plate cases. Thus, it was expected that the corresponding two and 3-D cases would give identical results with constant properties in the third direction.

The results have shown that all cases give identical final flow fields and essentially equivalent convergence histories. It can thus be said that the code is consistent in its treatment of boundary conditions and solution in all directions. It should also be noted that these cases were conducted using a four block domain formulation to test boundary conditions at block interfaces as well as domain boundaries. Thus, it can further be said that each block is treated consistently in the code.

Further simple tests include a block independence test which compares solutions of identical physical flow domains with a different number of blocks. This test, again, produced identical flow field solutions in all cases. The exact maintenance of freestream flow was also successfully tested.

3.7.2 Mach 2 Flat Plate

To further confirm the operation of the Navier-Stokes formulation a testcase was conducted using a Mach 2 flat plate. The case is presented in a paper by Lawrence et al.\textsuperscript{77} The uniform freestream flow conditions were: $M_{\infty} = 2$, $p_{\infty} = 2566$ Pa, $T_{\infty} = 221.6$ K, $T_{\text{wall}} = 221.6$ K, $\varphi = 0$ (air). The total grid size was $78 \times 23$ with a physical domain of $1.0 \times 0.01$ metres. A comparison of the normalized velocity and temperature profiles at the streamwise location $x = 0.915$ m for the present code and that of Lawrence et al. are shown in Figs. 3.2 - 3.3. Excellent agreement can be seen between the present code and the results computed by Lawrence et al.
The convergence residual for this case is shown in Fig. 3.4. Smooth and stable convergence is observed.

3.7.3 2-D 15° Hypersonic Compression Corner

The 15° Hypersonic Compression Corner validation case has been used by several previous researchers for code validation (see for example Ref. 77, 79–81). The experimental data used for comparison were taken from the work of Holden and Moselle. This data is widely accepted as reliable and very suitable for code validation. The test case consists of hypersonic flow over a two-dimensional compression corner. The flow is known to be completely laminar, thus it is independent of turbulence modeling issues and therefore well suited to test a laminar Navier-Stokes solver. A generalized schematic of the flow is given in Fig. 3.5. This figure reveals that the deceptively simple geometry produces quite a complex interaction between viscous and inviscid flow. The leading-edge shock can be seen due to the presence of a thick hypersonic boundary layer. This shock interacts with the resultant shock from the compression corner and produces an expansion fan which again interacts with the downstream boundary layer. In the vicinity of the base of the ramp there exists a region of recirculating flow for a certain range of Mach and Reynolds numbers and results from the the flow in the core of the boundary layer having insufficient momentum to overcome the adverse pressure gradient and skin friction. This region is absent for the relatively mild wedge angle of 15° in the present validation case but, as will be shown later, for increasing wedge angle the recirculation region becomes significantly large with a consequential increase in wall surface pressure. The boundary layer is seen to decrease dramatically in thickness after passing through the high pressure region behind the resultant shock. It thins to a minimum and gradually increases in thickness downstream due to the effect of the expansion fan.

The uniform freestream flow conditions for this case were: \( M_\infty = 14.1, \; Re/L = 2.4 \times 10^5 m^{-1}, \; p_\infty = 14.98 \; Pa, \; T_\infty = 88.89 \; K, \; T_{wall} = 297.2 \; K, \; \phi = 0 \) (air). The computational grid used in the present solution was 160 nodes in the streamwise (\( \xi \)) direction and 70 in the transverse (\( \eta \)) direction. The length of the flat plate portion of the domain was \( L = 0.439 \; m \) and the computational domain extended to 2L in the streamwise direction.

Mach number contours for the numerical solution produced by the present algorithm are shown in Fig. 3.6. The general features described schematically above are clearly visible.

Plots for the pressure coefficient, skin-friction coefficient and heat transfer coefficient along
the ramp wall are shown in Figs. 3.7 – 3.9 respectively. These quantities are calculated as

\[ C_p = \frac{2p}{\rho \alpha u_x^2} \]  
\[ C_f = \frac{2\tau_w}{\rho \alpha u_x^2} \]  
\[ C_h = \frac{\dot{q}_w}{\rho \alpha u_x (H_{\alpha} - H_w)} \]

where \( H_w, \tau_w \) and \( \dot{q}_w \) are the total enthalpy, shear stress and heat transfer rate at the wall respectively. Excellent agreement between the present results and the experimental measurements due to Holden and Moselle for all quantities can be seen. These results are also in excellent agreement with several well-known flow solvers utilized at NASA, which were also used to solve the present case in Ref. 80.

Finally, the convergence residual is shown in Fig. 3.10. Smooth and stable convergence toward machine zero is observed. While the number of iterations seems high at first blush, acceptable convergence is achieved after about four or five orders of magnitude reduction in convergence residual.

3.7.4 24° Hypersonic Compression Corner

A second compression corner validation case was examined with a compression angle of 24° with identical freestream flow conditions to the 15° case (see Sec. 3.7.3). This case is a very interesting and challenging problem for a flowsolver. The increase in compression corner angle causes a very large recirculation region for the present freestream flow conditions (see Fig. 3.5). The case is particularly interesting since it has been asserted (principally by Rudy et. al.) that the proper flow field solution is only realizable by considering three-dimensional effects. Rudy et. al. rather convincingly argues that the physically 3-D flow field has a component of "spillage" in the third dimension over the planar ramp used in the experiments of Holden et. al. which must be considered in the calculations. Evaluation of this flow field with a two-dimensional solver leads to an over-estimation of the recirculation region. This, together with the conclusion that a relatively fine grid must be employed, resolves the discrepancy some past researchers have had in solving this problem (see for example, Refs. 79, 81, 83, 84) using a two-dimensional flow solver. The results of the present study support the hypothesis and evidence advanced by Rudy et. al. and confirm their findings, i.e., agreement with experimental results can only be accomplished using a 3-D flow solver and considering the finite-span effects of the compression corner. Thus, this flow problem is a well suited validation case for a 3-D flow.
Solution of the present validation case was initially performed in two-dimensional computational space in order to confirm the prediction of an over-estimation of the recirculation region. It is proposed that similar trends for the 2-D solutions between the current code and the multiple codes tested by Rudy et al. lend further credibility to the proper operation of the current code. In order to rigorously determine the 2-D solution a limited grid convergence study was completed. Fig. 3.11 shows the results of pressure coefficient at the wall boundary for three grids: 100 x 50, 200 x 100 and 400 x 200. The over-estimation of the recirculation region, as evidenced by a sudden rise in $C_p$, is clearly visible for each grid. Further, the effects of grid refinement seem to produce results which converge asymptotically to a solution. The very small difference between the 200 x 100 grid and the 400 x 200 grid results suggest that this solution is grid converged. Therefore, it can be asserted with some confidence that solution of this flow field in two-dimensional computational space results in a large over-estimation of the recirculation region. These results are consistent with Ref. 80.

In order to investigate the finite-span effect, 3-D computations were carried out on several computational grids. The width of the compression corner ramp in the experiments was 0.609 m and no side plates were used to contain the flow. Numerically this was modeled with a spanwise length of 0.3 m in the third dimension with a symmetry boundary condition imposed at the ramp centre-plane. The flow field is shown schematically in Fig. 3.12. The computational domain extended along the (half) width of the ramp and partially into the adjacent freestream flow. Twenty five percent of the grid cells used in the spanwise ($z$) direction resided in the freestream for each case.

Figure 3.13 shows the pressure coefficient at the ramp wall along the centre-plane for 3-D computations for several grid systems. It can be seen that excellent agreement between experiment and numerical simulation exist for the finest computational grid employed and the large overestimation of the recirculation region realized in the 2-D computations is eliminated. It can also be seen that this case is numerically grid-converged given the close agreement between the two finest grid systems employed.

The skin-friction coefficient and heat transfer coefficient along the ramp wall at the centre-plane are shown in Figs. 3.14 and 3.15 for the finest (200 x 100 x 20) grid. Good agreement can be seen for these parameters as well.
Figure 3.2: Normalized velocity profiles at $x = 0.915$ m.

Figure 3.3: Normalized temperature profiles at $x = 0.915$ m.
FIGURE 3.4: Convergence history.
FIGURE 3.5: Schematic of viscous compression corner flow field.

FIGURE 3.6: Mach number contours for 15° compression corner validation case.
Figure 3.7: Pressure coefficient along the ramp wall for 15° compression corner validation case.

Figure 3.8: Skin friction coefficient along the ramp wall for 15° compression corner validation case.
Figure 3.9: Heat transfer coefficient along the ramp wall for 15° compression corner validation case.

Figure 3.10: Convergence residual for 15° compression corner validation case.
Figure 3.11: Pressure coefficient along ramp wall for several 2-D computational grids for 24° compression corner validation case.

Figure 3.12: Schematic of compression corner flow field.
Figure 3.13: Pressure coefficient along ramp wall centre-plane for several 3-D computational grids for 24° compression corner case.

Figure 3.14: Skin friction coefficient along ramp wall centre-plane for 3-D computations for 24° compression corner case.
Figure 3.15: Heat transfer coefficient along ramp wall centre-plane for 3-D computations for 24° compression corner case.
The results obtained through the numerical simulation of the ramp fuel injectors considered are presented in the present chapter. A review of the methodology used to analyze these results will be given followed by a presentation and discussion of said results.

4.1 Analysis of Numerical Results

The analysis of numerical injector flow field solutions consists of two main approaches: flow field analysis and performance parameter analysis. The dynamics of the flow will be elucidated via the flow field analysis and more quantitative evaluation of injector performance will be afforded by a number of integrated performance measures.

4.1.1 Flow Field Analysis

Flowfield visualization of three-dimensional solutions is a challenging undertaking due to the wealth of information contained within the solution and the inherently two-dimensional media (i.e. print) used to present the flow field. Two-dimensional plots representing cross-sections of a 3-D solution are used extensively, both individually and in series, to ascertain flow field characteristics. Careful interpretation of these plots is warranted due to the highly complex, three-dimensional nature of fuel/air mixing flow fields. Evolution of the mixing of hydrogen in the surrounding airflow is primarily addressed via analysis of transverse cross-sectional planes ($yz$) at a series of streamwise locations ($\bar{x}$) downstream of injection. Through such parameters
as fuel (H₂) mass fraction contours, pressure contours and velocity vectors, a picture of the flow field can be realized. Select longitudinal planes (xz) further contribute to an understanding and visualization of flow field dynamics.

4.1.2 Performance Parameters

A number of measures were calculated to address the mixing performance of the various ramp injector configurations in a quantitative context. These performance parameters are generally calculated as a function of non-dimensional streamwise distance and, where appropriate, integrated in the transverse (yz) plane and provide a means of comparison between injectors. A brief description of each performance measure, together with calculation details, is provided in the following subsections.

Circulation

The ramp fuel injectors presented in the current work introduce mixing augmentation primarily through the introduction of axial vortices. The strength and nature of these vortices are tantamount to the mixing performance of a given injector geometry. An integrated measure of the axial vorticity is provided by the circulation in a given transverse plane. Non-dimensional circulation is defined in the context of the present study as

\[ \Gamma_{yz} = \frac{1}{u_x \sqrt{A_f}} \int \left( \frac{\partial v}{\partial z} - \frac{\partial w}{\partial y} \right) dA_{yz} \]  

(4.1)

where \( u_x \) is the freestream air velocity and \( A_f \) is the projected fuel injection area. This measure must be interpreted with caution however. Vortices of opposite sense will have a counteractive contribution to the circulation measure. (Note that the circulation is calculated over the computational domain which consists of a symmetric half of one injector.) Further, circulation produced in areas remote to the fuel/air interface will have little if any mixing benefits.

Mixing Efficiency

A widely used measure of mixing performance is the mixing efficiency.\(^85\) It is defined as the fraction of injectant that could react if combustion occurred without further mixing. Formally, for a globally fuel rich mixture,

\[ \eta_m = \frac{\int Y_{H_2,R} \rho u dA_{yz}}{f \cdot \int (1 - Y_{H_2}) \rho u dA_{yz}} \]  

(4.2)
where

\[
Y_{H_2,R} = \begin{cases} 
Y_{H_2}, & Y_{H_2} \leq Y_{H_2}^\text{st} \\
Y_{H_2}^\text{st} \left(1-Y_{H_2} \right), & Y_{H_2} > Y_{H_2}^\text{st}
\end{cases}
\] (4.3)

and \(Y_{H_2}^\text{st}\) is the stoichiometric hydrogen mass fraction and \(f\) is the stoichiometric fuel/air mass ratio. Thus, in regions of the flow where the local mixture is fuel lean (\(\varphi < 1\), i.e. \(Y_{H_2} < Y_{H_2}^\text{st}\)), the entire quantity of fuel can react, while in locally fuel rich mixtures (\(\varphi > 1\)), reaction occurs to the extent of available oxygen. Mixing efficiency can lend insight into the potential combustion performance of a given configuration but does not account for the coupled influence of combustion on the mixing process.

**Mixedness Measure**

A mixing performance parameter, first introduced by Waitz et al.,\(^{43}\) compares the fraction of total fuel mass flux present at various mass fractions at each axial station,

\[
\frac{\dot{m}_{H_2}}{\dot{m}_{H_2,\text{total}}} \text{ vs. } Y_{H_2} \text{ vs. } \bar{x}
\]

This so-called, mixedness measure, reveals the amount of fuel, \(H_2\), (as a fraction of the total fuel mass flux) that is present at various mass fractions in a given transverse plane as a function of axial, or downstream, distance. Plots generated for this parameter take the form of three-dimensional carpet plots and reveal a large amount of information relating to the suitability of the fuel/air mixture for combustion. In order to achieve efficient combustion, the fuel and air must be mixed, ideally, to stoichiometric proportions. In the case of hydrogen fuel this translates to a fuel mass fraction of approximately 2.8%. Thus, it is advantageous to have the entire cross-section, at a given distance downstream of injection, as close as possible to stoichiometric proportions. This manifests itself in the context of the mixedness measure as a shift of the fuel (fraction of total fuel mass flux) toward lower fuel mass fractions (ideally \(Y_{H_2} \leq 0.028\)) or equivalently, a shift of the volume under the curve toward lower fuel mass fractions.

**Fuel Jet Penetration Trajectory**

The ability of the fuel jet to penetrate the surrounding airstream is of importance in hypersonic propulsion applications. The high-temperature, low-momentum flow regime near solid surfaces (especially near the lower wall) must be avoided by the hydrogen jet. For shock-induced
combustion ramjets this would lead to premature ignition while in scramjet applications excessive heat release near engine surfaces leads to excessive cooling loads and materials problems. A measure of the fuel jet penetration trajectory is defined by the streamwise tracking of the fuel jet centre of mass. The \( z \) coordinate of this trajectory is presented as a function of non-dimensional streamwise distance \( (\bar{x}) \) and calculated on a discrete mesh as

\[
\bar{z}^* = \frac{\sum_{j,k} m_{H_2,j,k} \bar{z}_{j,k}}{\sum_{j,k} m_{H_2,j,k}}
\]

(4.4)

where \( m_{H_2,j,k} \) is the fuel mass flow rate at a given point in the \( yz \) plane.

**Decay of Maximum Fuel Mass Fraction**

A simple and often used measure of mixing performance is the decay of maximum fuel mass fraction. This measure tracks the maximum fuel mass fraction as a function of streamwise distance downstream of injection. While rapid decrease of this measure after injection can indicate beneficial mixing performance, the distribution of the fuel mass fractions must also be considered (which is clearly revealed in the mixedness measure; see above).

**Transverse Fuel Jet Area**

Stretching of the interfacial area between the fuel jet and surrounding air is a primary mechanism in increasing molecular mixing via diffusion. A measure of this interfacial area can be inferred from the extent of the fuel area in the projected transverse plane. The transverse fuel jet area is defined as the area in the \( yz \) plane contained within the \( Y_{H_2} = 0.005 \) fuel mass fraction. This area is non-dimensionalized by the initial fuel jet injection area, \( A_f \).

**Total Pressure Losses**

Increases in mixing performance must be analyzed in the context of the losses incurred to facilitate such gains. Improved mixing performance which come at the cost of significant flow losses may not be suitable for practical implementation. The concept of loss-effective mixing can be interpreted as generation of flow losses which contribute only to mixing effectiveness. All mixing processes will increase losses due to diffusion. While these losses are mitigated by potentially increased propulsive performance, losses that are not attributed to improved mixing are not redeemable and should be avoided. Given that it is very difficult to classify losses as mitigated or not, only bulk flow losses are calculated. A measure of the flow losses
used in propulsion applications is the reduction in total pressure. Severe total pressure losses have significant effects on the efficiencies of a hypersonic propulsion vehicle. Total pressure reduction can be caused by shock waves, viscous and shear forces in the boundary layer, flow separation and fuel/air mixing. A mass averaged total pressure loss parameter may be defined by

$$\Pi = \frac{\int P_0 \rho u dA_{yz}}{\int P_{\infty} \rho u dA_{yz}}$$

(4.5)

This measure relates the integrated mass-averaged total pressure in a given transverse plane to the initial total pressure and is presented as a function of non-dimensional streamwise distance.

<table>
<thead>
<tr>
<th>Case</th>
<th>Description</th>
<th>$\alpha_c$</th>
<th>$\alpha_e$</th>
<th>$\alpha_s$</th>
<th>$h$ [mm]</th>
<th>$w$ [mm]</th>
<th>$A_f$ [mm$^2$]</th>
<th>$L$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>Cantilevered (unswept)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>A2</td>
<td>Conventional (unswept)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>A3</td>
<td>Cantilevered (swept)</td>
<td>7°</td>
<td>-7°</td>
<td>5°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>A4</td>
<td>Conventional (swept)</td>
<td>7°</td>
<td>-7°</td>
<td>5°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>B1</td>
<td>Cantilevered (unswept)</td>
<td>0°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>318</td>
</tr>
<tr>
<td>B2</td>
<td>Conventional (unswept)</td>
<td>0°</td>
<td>-7°</td>
<td>0°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>318</td>
</tr>
<tr>
<td>C1</td>
<td>Cantilevered (unswept)</td>
<td>7°</td>
<td>0°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>318</td>
</tr>
<tr>
<td>C2</td>
<td>Conventional (unswept)</td>
<td>7°</td>
<td>0°</td>
<td>0°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>318</td>
</tr>
<tr>
<td>D1</td>
<td>Cantilevered (opposing)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>E1</td>
<td>Cantilevered (extended)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>175</td>
</tr>
<tr>
<td>BL</td>
<td>Incident boundary layer</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BL-A1</td>
<td>Cantilevered (unswept)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>FF</td>
<td>Far-field study</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FF-A1</td>
<td>Cantilevered (unswept)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>FF-A2</td>
<td>Conventional (unswept)</td>
<td>7°</td>
<td>-7°</td>
<td>0°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>FF-A3</td>
<td>Cantilevered (swept)</td>
<td>7°</td>
<td>-7°</td>
<td>5°</td>
<td>20</td>
<td>20</td>
<td>400</td>
<td>159</td>
</tr>
<tr>
<td>FF-A4</td>
<td>Conventional (swept)</td>
<td>7°</td>
<td>-7°</td>
<td>5°</td>
<td>40</td>
<td>10</td>
<td>400</td>
<td>159</td>
</tr>
</tbody>
</table>

Table 4.1: Computational cases considered.
4.2 Injector Configurations

The ramp fuel injector configurations presented in the test matrix of Sec. 2.1.2 are summarized in the present section with emphasis on the format of presented results. Some of the germane features and geometric parameters associated with the injector configurations considered are recapitulated in Table. 4.1.

4.2.1 Baseline Injector Configurations

The baseline injector configuration for the present study is designated as Configuration A1: unswept cantilevered ramp injector with $\alpha_c = 7^\circ$, $\alpha_c = -7^\circ$ and zero sweep angle. This configuration will serve as a reference to which various configurations can be compared with respect to mixing performance. A detailed review of the flow field features of this configuration will serve to illuminate the characteristics of a cantilevered ramp fuel injector.

In order to assess the mixing performance of the cantilevered ramp fuel injector in the context of existing designs, a baseline conventional configuration (A2) was specified with similar geometric attributes as the baseline cantilevered ramp injector.

4.2.2 Parametric Study of Injector Geometry Parameters

Various injector configurations were investigated in order to lend insight into the flow field and mixing characteristics of the cantilevered ramp injector. Results are given which highlight the unique flow field features as compared with the baseline configuration. In several cases, comparison of a cantilevered ramp fuel injector configuration is made with a conventional injector counterpart.

4.2.3 Grid Convergence Study

Grid convergence of all flow field features in a three dimensional simulation is practically very difficult given contemporary computational power. Employing finer and finer grids until flow features and parameters remain unchanged is a common method of determining grid independence in two dimensions but this luxury is not afforded in three dimensions due to the tangible restrictions on computational speed and memory which are rapidly consumed for large 3-D grids. Nevertheless, issues of solution dependence on the computational mesh must be addressed. A limited grid convergence study was conducted in an effort to show that the computational grid used was adequate to capture the essential flow field features of interest. A number of grid systems were employed for the same computational domain to assess grid
convergence. Using the grid system employed for subsequent results as a baseline to compare other systems, grids of 12%, 30%, 50%, 75% and 140% of the number of cells in the baseline were numerically solved given identical injector conditions and domains. The injector geometry selected for the grid convergence study was Configuration A1 as defined in Sec. 2.1.2. Qualitative comparison of the baseline grid with the finer grid (140%) showed good agreement in injected fuel structure and distribution (a primary interest for mixing studies), while much more coarse grids did not adequately resolve features of interest. Quantitatively, these grid systems were compared using the mixing efficiency previously defined (see Sec. 4.1.2). Figure 4.1a plots the mixing efficiency as a function of downstream distance for the various grid systems. Figure 4.1b shows the values of mixing efficiency at \( x = 10 \) against grid size. The close agreement between the results of the primary grid and the fine grid give confidence in the results of the baseline grid system used. Similar grid densities and distributions were employed for the remaining injector configurations.

4.3 Discussion and Interpretation of Results

The converged numerical flow fields are presented in the current section employing the analysis strategy outlined in Sec. 4.1. These results are evaluated and interpreted in the context of fuel/air mixing performance. Further discussion and more comprehensive comparative conclusions are given in Chapter 5.
4.3.1 Baseline Cantilevered Ramp Injector Configuration

A succinct picture of the mixing evolution in the baseline injector flow field is provided by Fig. 4.2, which shows the hydrogen mass fraction contours at select transverse planes downstream of injection. Also shown in this figure are the shaded surfaces of a single injector unit (note that data has been mirrored in the $xz$ symmetry plane of the injector for clarity of presentation). Figure 4.2 also serves to give an overview of the computational domain of a single injector flow field ($SD_1$ and $SD_2$ as discussed in Sec. 2.2.3). The most striking feature of the flow field is the distortion and stretching of the fuel/air interface by longitudinal vortices with the attendant increase in interfacial area. The general metamorphosis of the initially square cross-section of the hydrogen jet to a “mushroom” shape and finally into two oblong segments is evident as the hydrogen traverses the computational domain in the streamwise direction. The driving force behind this transformation is the generation of streamwise vorticity. Specifically, as will be presented shortly, a dominant pair of counter-rotating vortices are generated by the cantilevered injector.

More detail of the injected hydrogen structure is given by Fig. 4.3 which displays hydrogen mass fraction contours at transverse planes near the injection plane ($\bar{x} = 0$). An immediate expansion of the fuel jet, toward the bottom wall, into the surrounding airflow is noted. This is primarily caused by the low pressure region created under the cantilevered ramp injector as the flow passes through a series of expansion waves. Also apparent is the “roll-up” present at the lower surface of the mass fraction contours. This can be interpreted as entrainment of air into the fuel jet. This entrainment is seen to continue and intensify as the fuel travels downstream. This entrainment is clear evidence of large-scale bulk mixing of the fuel and air and is a precursor to enhanced mixing due to the increase in contact area between the fuel and air. Indeed, at $\bar{x} = 10$ it can be seen that the interfacial area between the fuel and air is approximately double its initial value. As will be shown subsequently, the distortion in mass fraction contours, in the injector near-field ($\bar{x} < 6$ in this case), at the upper corners is due mainly to vorticity created by the cross-stream shear mechanism while the more pronounced distortion near the bottom of the contours is, in part, due to vorticity created by baroclinic torque effects.

Still more detail of the fuel jet behaviour after injection is furnished by Fig. 4.4. Hydrogen mass fractions at select transverse planes (specifically: $\bar{x} = 0.1, 2, 4, 6, 8, 10, 15, 20, 25, 30, 35, 40$) are shown in the range $0.05 < Y_{H_2} < 1.0$ with a contour interval of $0.05$ (note that the same range is used for all subsequent injector plots facilitating consistent comparison). Qualitative evidence of bulk mixing between fuel and air is demonstrated in the series of plots. In addition to the stretching and contortion of the fuel/air interface, diffusion of the hydrogen is evident
with increasing downstream distance; note the maximum mass fraction contour at $\bar{x} = 40$ is $Y_{H_2} = 0.5$ while the full range of contours exist in earlier plots.

Insight into the characteristic behaviour of the hydrogen fuel jet as described above can be gained by inspection of the complex pressure and velocity distributions present in the flow field. The static pressure and velocity vectors are shown in Figs. 4.5 and 4.6 at the previously mentioned transverse planes. Inspection of the pressure contours reveals complex, highly three-dimension flow structures and discontinuities. The shocks shown schematically in Fig. 1.2 can be seen as close clustering of pressure contours in Fig. 4.5. The shock wave generated by the top compression surface of the ramp injector (labeled Shock B in Fig. 1.2) is represented as a strong curving shock wave in the $\bar{x} = 0.1$ plot and its initial reflection at the top wall boundary is seen at $\bar{x} = 4$. Similarly, the shock generated at the base of the expansion trough (labeled Shock A in Fig. 1.2) can be seen in the lower portion of the plots just downstream of injection. This shock plays a key role in the generation of baroclinic torque vorticity. Multiple shock reflections are evident with generally decreasing pressure gradients with increasing streamwise distance. Of interest is the highly curved shock structure evident near the centre of the plots at $\bar{x} = 6, 8$ and 10. This pressure gradient is formed by the large vortex structures (to be shown subsequently) penetrating up into the freestream flow. The strength of these structures is evidenced by the resultant pressure gradients.

Velocity vectors for the corresponding transverse planes are given in Fig. 4.6. The large vortex structures alluded to in the previous representations of the flow field solution are clearly observed. Strong counter-rotating vortices are already present in the injection plane as shown in the plot corresponding to $\bar{x} = 0.1$. These vortices are, in the main, produced by the cavity located under the cantilevered injector. The strength of these vortices is appreciable, and indeed, constitutes the distinguishing flow characteristic of a cantilevered fuel injector (which will be made more clear in a subsequent comparison with a conventional injector). Moving downstream of fuel injection, the expansion of the hydrogen fuel toward the lower wall is evident at $\bar{x} = 0.1$ and more clearly at $\bar{x} = 2$. This expansion interacts with the vortices created under the fuel injector, mitigating their effect to some degree largely by forcing the vortices to interact with the lower wall. At the downstream distance of $\bar{x} = 4$, the shock anchored at the base of the expansion trough is distinguishable by a sudden turn of the velocity vectors along a line near the bottom of the plot. This shock can be tracked in subsequent plots downstream and is shown to intensify and broaden (see especially $\bar{x} = 10$) the vortices via the baroclinic torque effect. It is clear that the baroclinic torque mechanism serves to contribute synergistically to the axial vorticity in the flow field producing vorticity of the same sense as those previously established by cross-stream shear generation. At streamwise distances more remote from the fuel injection.
plane (i.e. \( \bar{x} > 20 \)) a marked migration of the primary vortices (and hydrogen fuel) toward the upper wall is discerned. This penetration into the main flow can be inferred qualitatively as a beneficial effect due to the higher momentum and lower temperatures found remote from the wall boundaries.

A pictorial representation of the hydrogen mass fraction contour evolution is displayed in Fig. 4.7 in the form of carpet plots at the aforementioned transverse planes. Two spatial coordinates are plotted (\( \bar{y} \) and \( \bar{z} \)) with the third axis representing the hydrogen mass fraction. Just downstream of the fuel injection plane (\( \bar{x} = 0.1 \)) a segregated fuel distribution is readily apparent with most of the fuel existing in its pure (\( Y_{H_2} = 1.0 \)) form confined around the initial projected injection area (note that an ideally mixed fuel profile would manifest itself as a flat plot at a constant value of \( Y_{H_2} = Y_{H_2^{stochio}} = 0.0283 \) in the current contour representation). The dramatic effect of the axial vorticity in the flow field in reducing and spreading the mass fraction contours is evident. Notable are the two “columns” of fuel distinct at \( \bar{x} > 15 \) which represent regions of high fuel concentrations. These regions correspond to the regions occupied by the primary vortex structures. The stability of these structures in the far-field maintains high fuel mass fractions near their center.

Quantitative analysis for the baseline cantilevered injector configuration, through plots of performance parameters, will be discussed in the context of comparison to a corresponding conventional ramp injector design in the following section.

**Comparison with Conventional Ramp Injector**

Comparison of mixing performance of the baseline cantilevered ramp injector configuration (A1) is provided by Configuration A2 which dictates the same design angles in a conventional design. Figure 4.8 portrays the shaded injector geometry of the unswept conventional ramp configuration with hydrogen mass fraction contours in select transverse planes. Further contour plots of fuel mass fraction for this configuration is shown in Fig. 4.9. A metamorphosis of the fuel contours is witnessed similar to that of the baseline configuration (A1). Air is entrained from below the fuel jet eventually forming a similar lobed shape downstream. Distortion of fuel contours due to both cross-stream shear vorticity and baroclinic torque generated vorticity is also evident in the injector near-field. Significant differences are revealed however comparing the plots of fuel mass fraction for both configurations (Figs. 4.2 – 4.4 for Configuration A1 and Figs. 4.8 – 4.9 for Configuration A2). Generally greater stretching and distortion of the fuel/air interface is demonstrated for the cantilevered injector (A1). This may be interpreted as qualitatively superior mixing performance for the cantilevered design over its conventional counterpart. This is clearly seen by comparing the plots at \( \bar{x} = 10 \) where the contour lines
are significantly longer for the cantilevered case, indicating increased interfacial area. These plots would also indicate greater entrainment of air into the fuel jet caused by increased vortex strength. Further downstream of injection, a greater spread of contours and diffusion of gradients is revealed in the case of the cantilevered design. Differences in hydrogen jet behavior are easily visualized by comparison of fuel mass fraction carpet plots (Figs. 4.7 and 4.12 respectively). More rapid reduction of high mass fractions and increased transverse extent are discerned for the baseline cantilevered injector configuration. More complete entrainment of air into the fuel stream is clear at, for example, $\bar{x} = 10$.

Pressure contours for Configuration A2 are given in Fig. 4.10. When compared to the static pressure contours for the baseline configuration (A1), the most significant difference is the more contiguous shock structure initiated at the base of the expansion trough in the case of the cantilevered ramp injector (shown in the lower portion of the near-field plots). This is tantamount to earlier and increased deposition of baroclinic torque for the cantilevered ramp injector. The corresponding shock in the case of the conventional configuration (A2) is rather weak and discontinuous in the lateral extent of the injected fuel. A stronger shock wave over the compression ramp surface is formed in the case of the cantilevered injector over its conventional counterpart. This is due to the larger projected area of the injector in the $xy$ plane. The effects of this more prominent shock on mixing performance are both positive and negative. The increased pressure created on the top surface of the ramp provides for an increased gradient for generation of cross-stream shear vorticity; conversely, the reflection of this strong shock wave from the top wall of the mixing duct imparts an adverse pressure gradient on the fuel/air density gradient which tends to form vorticity, via baroclinic torque, of an opposite sense to that created upstream.

A series of plots depicting the velocity vectors at various transverse planes is contained within Fig. 4.11 for Configuration A2. The most remarkable aspect of the velocity field for the conventional design is the lack of large-scale vorticity immediately downstream of fuel injection at $\bar{x} = 0.1$. This is to be expected due to the relative lack of vorticity generating mechanisms in this region but provides a noteworthy contrast to the significant vorticity evident in the baseline cantilevered configuration (see Fig. 4.6). Generation of vorticity due to baroclinic torque is evident in the plots of $\bar{x} < 15$ where the shock wave is seen to intersect the injected fuel jet from below. Further downstream, a counter-rotating pair of vortices is created similar to the baseline configuration with generally stronger vorticity (as evidence by increased magnitudes of velocity) present for the cantilevered ramp configuration (A1).

Comparison of the mixing performance parameters defined in Sec. 4.1.2, for the baseline cantilevered injector (A1) and its conventional counterpart (A2), is provided by Fig. 4.13. All
measures are presented as a function of non-dimensional streamwise distance ($\bar{x}$).

It is shown in Fig. 4.13a that the cantilevered ramp injector produces significantly more circulation than the conventional ramp injector upstream of fuel injection ($\bar{x} < 0$). This can be directly attributed to the strong vortices generated in the cavity under the cantilevered injector (see also Fig. 4.6). The sharp decline in circulation is a consequence of the expansion of the fuel jet down into the airflow which interferes with the vortices created under the injector, and forces them toward the lower solid surface. Interaction of the main vortices with the lower wall creates additional vortices of opposite sense which decrease the circulation measure. Downstream of injection ($\bar{x} > 4$) alternating increases and decreases in circulation are noted. This behaviour is largely governed by the contribution of baroclinic torque to axial vorticity. Increases in circulation correspond to intersection of the injected fuel jet with “favourable” pressure gradients (provided by shocks initiated at the lower solid surface), while decreases are the result of “adverse” pressure gradients produced by shocks reflected from the top surface of the computational domain. Because of the early formation and strength of the shock anchored at the base of the expansion trough (labeled Shock A in Fig. 1.2) in the cantilevered ramp case, the baroclinic torque is significantly stronger attaining a local maximum at $\bar{x} \approx 10$, whereas the corresponding local circulation maximum for the conventional ramp injector is appreciably lower and occurs at $\bar{x} \approx 15$. Indeed, the dramatic rise in circulation for the baseline cantilevered configuration in the range $2 < \bar{x} < 10$ is strong testimony to effective and significant vorticity generation via baroclinic torque. Subsequent fluctuations in circulation correspond to reflected shock intersections with the hydrogen/air interface. Generally, larger fluctuations are observed for the baseline cantilevered configuration (A1) due to the stronger pressure gradients present in the flow field (compare Figs. 4.5 and 4.10).

Computed values of mixing efficiency, for the baseline cantilevered and conventional ramp injectors are plotted in Fig. 4.13b as a function of streamwise distance. The cantilevered ramp injector design displays superior values of this parameter over the corresponding conventional configuration. At $\bar{x} = 40$ its mixing efficiency is approximately 48% higher than that of the conventional ramp injector. Superior mixing performance of the cantilevered design results from an increase in axial vorticity as demonstrated above.

Qualitative observation of an increased transverse dissipation of hydrogen downstream of injection in the case of the baseline cantilevered ramp injector is quantitatively reflected in Fig. 4.13c which plots the relative fuel area for both injectors considered. A generally increasing trend can be seen for both injectors with consistently higher values for the cantilevered ramp injector over its conventional ramp counterpart. A larger fuel jet area is indicative of a larger fuel/air contact surface which, in turn, is conducive to increased micro-mixing via dif-
fusion. The slight decline in relative fuel jet area for both injectors (witnessed in the range $5 < \bar{x} < 14$) is caused by the compression of the fuel jet from the vortices (strong in the cantilevered ramp injector case) produced under the fuel jet and moving up toward it. At $\bar{x} = 40$ the relative fuel jet area is approximately one third larger for Configuration A1.

The fuel jet penetration trajectory for both configurations is shown in Fig. 4.13d. Similar trends are observed for both cantilevered and conventional configurations. Noteworthy is the migration of the fuel jet toward the lower wall boundary just downstream of the injection plane in the case of the cantilevered ramp case. As noted earlier, the complex pressure field is primarily responsible for this behaviour, specifically, the low pressure region created under the cantilevered injector is responsible for the sharp downturn in fuel trajectory evident in the range $0 < \bar{x} < 8$. Pressure distributions within the flow fields suppress fuel penetration for both injectors until $\bar{x} \approx 25$ where favourable pressure conditions and vortex migration promote fuel penetration into the airstream.

The maximum mass fraction of hydrogen, in a given transverse plane, for both injectors as a function of streamwise distance is given in Fig. 4.13e. Ever decreasing values are noted for both plots with a significantly more rapid decline for the cantilevered configuration for $\bar{x} < 20$. An inflection point for Configuration A1 is clearly noted. This results from the separation and stabilization of the two main vortices in the flow field (see Fig. 4.6). Small regions of high mass fraction hydrogen are contained near the center of these vortices effectively “trapping” a small portion of high fuel concentration fluid in a region of lower convective flux. These regions are clearly discernible as two peaks in the plots of Fig. 4.7. The alignment of the centers of high mass fraction and vortex centers for the cantilevered case at $\bar{x} > 20$ shifts the decay of maximum fuel mass fraction from a convection driven process to a diffusion dominated process. This should not be interpreted however as a sudden decline in the mixing performance of the cantilevered injector as other performance parameters do not substantiate this claim and it is noted that the average mass fraction continues to decline smoothly over the streamwise domain.

The mass-averaged total pressure is plotted in Fig. 4.13f. Higher total pressure losses are demonstrated for the cantilevered configuration. Increased shock strength and shear stresses primarily account for this difference. At a downstream distance of $\bar{x} = 40$, a 15% decrease in total pressure recovery is noted for the cantilevered injector.

A comprehensive view of relative injector mixing performance is provided by the mixedness measure shown in Fig. 4.14. This measure was defined in Sec. 4.1.2. Comparison of the unswept cantilevered (A1) and unswept conventional configuration (A2) yields significant differences. The downstream distance required to eliminate all regions of pure hydrogen ($Y_{H_2} = 1.0$) is seen to be significantly shorter for the cantilevered design as previously men-
tioned. Further, a larger and earlier shift of the hydrogen mass toward the lower mass fractions is noted for the cantilevered case. This is indicative of improved mixing performance as lower hydrogen mass fractions imply increased air mass fractions, which are closer to optimal stoichiometric proportions. While the decay of maximum mass fraction was seen to abate further downstream for the cantilevered design, the mixedness measure shows a clear cascading of hydrogen toward the lower mass fractions continuously along the streamwise domain (shown as an increase in height of the plot at lower $H_2$ mass fractions). This is indicative of an ever decreasing average mass fraction which, in turn, implies increased diffusion and mixing (since a completely and optimally mixed flow field will comprise of a constant fuel mass fraction of $Y_{H_2} = Y_{H_2, stoichiometric} = 0.028$). A quantitative analysis of mixing performance is afforded by the percentage of fuel mass fraction mixed to levels below $Y_{H_2} < 0.05$ (this is indicated by the height of the edge of the carpet plot along the streamwise coordinate). Fuel and air mixed to this level promotes combustion for a reacting system. From Fig. 4.14 it can be seen at $\bar{x} = 40$, 15% of the fuel is mixed to this threshold for the cantilevered case (A1) whereas only 9% is mixed for the conventional case (A2).

### 4.3.2 Swept Cantilevered Ramp Injector Configuration

Having established a conception of the characteristics of the baseline cantilevered injector, and its performance compared to a corresponding conventional design, variations of the baseline configuration (A1) are investigated to elicit the effects of changes in injector geometry. As one of the most widely considered geometry changes to improve ramp injector mixing performance is to sweep the side walls of the injector, the effect of a non-zero sweep angle was investigated for the cantilevered ramp injector concept.

Insight into the mixing characteristics of the swept cantilevered ramp configuration is facilitated by plots of fuel mass fraction given in Figs. 4.15 and 4.16. Similar downstream evolution of the injected fuel to the baseline (unswept) injector is realized. Considerable entrainment of air into the hydrogen jet, mainly from below, is shown. Comparing the fuel contour shapes of the swept cantilevered injector (A3) with the baseline (A1) injector, a marked increase in transverse expanse is noted for the swept case. This is most notable in the region near the injection plane at $\bar{x} < 10$. A marked increase in vortices generated via cross-stream shear (which is a main characteristic of the swept design) is responsible for this behaviour. The stretching of the mass fraction contours in the $y$ direction is accentuated in the upper portion of the domain. As will be seen more clearly shortly, this is the region in which the cross-stream shear generated vortices exist. Also of interest in the plots of hydrogen mass fraction is the interaction of the fuel jet with the lower wall most visible in the range $20 < \bar{x} < 25$. Complex pressure distri-
butions in the flow field are responsible for this situation. The result of this interaction is a deposition of fuel along the lower wall and injector symmetry plane. This results in deleterious effects on mixing performance because it translates a significant portion of hydrogen mass into regions of low vorticity. Indeed, the fuel which is remnant near the lower wall is subject to low momentum which detracts from mixing performance. The fuel present in these regions is graphically shown in Fig. 4.19.

Further insight into the flow field of the swept cantilevered injector flow field described above is provided by plots of static pressure contours and velocity vectors in the corresponding transverse planes; these plots are shown in Figs. 4.17 and 4.18. Pressure contours convey the main shock structures of the flow field. The shock generated by the compression surface of the ramp is clearly visible in the first two plots as a strong almost planar (uniform in the spanwise direction) discontinuity. The larger projected area of the compression ramp (produced by side-wall sweep) accounts for this. The pressure gradient primarily responsible for baroclinic torque generated vorticity in the injector near-field is evident in the lower portions of the graphs for $\bar{x} < 10$. Generally, strong shocks are noticeable throughout the streamwise domain. Conspicuously these shocks remain strong and pronounced at large downstream distances ($\bar{x} > 35$).

Velocity vectors for this configuration reflect intense transverse velocity components and vorticity. The large counter-rotating vortex pair characteristic of the cantilevered design are formed and intensified downstream. The first plot also reveals vortices due to the cross-stream shear generation mechanism near the injector side walls (see further discussion below). Complex velocity patterns in the far-field are a result of strong pressure gradients which do not allow stabilization of vortices. Reverse pressure gradients generate conflicting vorticity of opposite sense (note that the regions above the shock in plots $\bar{x} = 35$ and $\bar{x} = 40$ have a higher pressure than those below, producing a gradient in the opposite direction to that in the injector near-field).

Comparison with Baseline Cantilevered Ramp Injector

Additional appreciation of the dynamics of the cantilevered ramp injector is allowed by comparison of select flow field parameters for the unswept baseline (A1) and swept configuration (A3). Figures 4.20 and 4.21 show the pressure contours of both injector cases with the stoichiometric $H_2$ mass fraction contour superimposed for the planes $\bar{\rho} = 0$ and $\bar{\rho} = 0.5$ respectively. The mass fraction contour gives an approximate idea of the domain extent of the hydrogen jet (in the given plane; note, these are not streamlines). Note that the scale of the abscissa and ordinate in these plots is independent to facilitate compact presentation of data.

A revealing feature in these plots is the increased intensity, and indeed, the dominance of
the shock generated by the ramp compression surface in the swept configuration. The shock anchored at the base of the expansion trough in the fuel injection plane ($\bar{x} = 0$) is shown to be of comparable strength and structure between injector configurations. These features are the fundamental reason behind the performance characteristics to be discussed subsequently. Positive and negative contributions of baroclinically generated vorticity are qualitatively portrayed as an intersection of the fuel jet (or more precisely, the fuel/air density gradient at the interface) with shock waves. The region close to the injector, at $\bar{x} < 10$, shows a very similar pressure field encountered by the fuel jet for both configurations. As the shock created by the compression ramp surface reflects from the top domain wall and intersects first the shock from the base of the trough and then the hydrogen fuel, a pressure gradient of opposite direction is realized and vorticity is generated of opposite sense. It is clear that, in the case of the swept configuration, a more substantial pressure gradient is imposed on the density gradient of the fuel interface (see the strong shock centered around $\bar{x} \approx 15$). Also note the deflection of the mass fraction contour which proves to reduce the extent of the fuel jet in the $z$ direction. Subsequent reflection of this shock from the bottom wall imposes a further positive contribution to the vorticity while a second reflection from the upper wall imparts baroclinic torque in the opposite (negative) direction. The remarkable strength of this second reflected shock for the swept configuration generates appreciable vorticity serving to mitigate the “positive” vorticity imparted on the injected fuel upstream. A marked deflection of the fuel mass fraction contour is again seen. Further, as alluded to earlier, the more pronounced pressure gradients in the swept case drive the injected fuel jet into contact with the lower wall. This is clearly evident in these figures as an intersection of the mass fraction contour with the lower wall (see Figs. 4.20b and 4.21b).

Detailed velocity vectors in the transverse injection plane ($\bar{x} = 0$) are shown for several configurations in Fig. 4.22. Comparison of the baseline configuration (A1) and the swept cantilevered configuration (A3), shows appreciably more vortices created near the injector side walls for the swept case. In addition, the velocity vectors adjacent to the injector have a more favourable orientation for promotion of axial vortices in the swept configuration. Similar vortex structures and intensities are displayed below the injector with slightly larger secondary vortices at the lower solid surface for the swept case. These vortices detract from the circulation measure in the flow field and have limited if any fuel/air mixing benefit. The location of the vorticity in both configurations support the supposition that the cross-stream shear vortices act primarily on the top portion of the fuel profile while the vortices generated under the injector and those subsequently produced via baroclinic torque act on the fuel jet from below (in the injector near-field).
Comparison with Conventional Ramp Injector

Comparison of the swept cantilevered ramp injector configuration (A3) is provided by its counterpart conventional design, designated Configuration A4. Hydrogen mass fraction contours together with shaded injector surfaces are displayed for this configuration in Fig. 4.23. More detailed plots of fuel distribution are given in Fig. 4.24. Similar contortions of the fuel profile compared to the swept cantilevered design is seen. Generally, a greater expanse of fuel in the spanwise (y) direction is shown for the cantilevered design. A notable difference for the case of the conventional design is the absence of interaction of the fuel jet with the lower wall. This is due to the changes in flow field pressure distribution (see Fig. 4.25). A smaller projected area of the top compression surface of the conventional design produces less intense shocks which, in turn, do not force the fuel toward the wall to the same degree. Further, the low pressure region created under the injector in the case of the cantilevered configuration is absent for the present conventional design.

Velocity vectors for the conventional ramp injector design are given in Fig. 4.26. Additional vorticity due to ramp sweep and increased cross-stream shear generation are noted in the injector near-field. Velocity components culminate into a pair of stable counter-rotating vortices downstream of injection. The lack of strong reflected shocks in the conventional design produces smaller amounts of “negative” vorticity (vorticity opposite in sense to the primary vortices) and consequently, little disruption to the primary vortices. A closer look at the velocity vectors at the fuel injection plane is shown in Fig. 4.22. Scrutinization of the swept cantilevered (A3) plot and swept conventional (A4) plot reveals interesting differences. Similar vorticity is indicated for both configurations along the ramp side walls. Dramatic additional vorticity generation is evident for the cantilevered case. The vortices generated under the cantilevered injector provide for substantially more axial circulation at the injection plane. Similar conclusions are drawn comparing the unswept cantilevered and conventional designs (A1 vs. A2). One of the main design motivations of the cantilevered design was the generation of these additional vortices. Comparison of plots in Fig. 4.22 bear out the effectiveness of this design concept in producing these vortices.

Figure 4.28 compares a series of performance parameters for the swept cantilevered injector and the swept conventional injector configuration. Focusing on the plot of circulation (Fig. 4.28a), a sharp rise is apparent for both injectors upstream of the injection plane ($\bar{x} = 0$) with a steeper increase for the cantilevered case afforded by the additional vorticity created by the cavity under the cantilevered injector (primarily in the region $-4 < \bar{x} < 0$). As with the baseline configuration, the swept cantilevered configuration (A3) realizes a sizable decline in circulation just downstream of injection (due to fuel expansion and interaction with the lower
The primary increase for both curves in the injector near-field is attributed to baroclinic torque deposition. Again, the performance of the cantilevered ramp injector, insofar as imparting more circulation on the flow field, is appreciably superior to its analogous conventional design. Indeed, the swept cantilevered configuration increases the circulation by over a factor of two during primary baroclinic torque generation in the streamwise range $2 < \tilde{x} < 8$. The decline in circulation for both curves in the approximate range $8 < \tilde{x} < 20$ is due to the counteracting baroclinic torque brought about by shocks reflected from the top surface. Previous presentation of more qualitative evidence of this is supported quantitatively by the integrated circulation measure. The increased strength of the reflected shock in the case of the swept cantilevered configuration (see discussion above) corresponds to a larger decline in the circulation curve, to the extent that the circulation is reduced to the level of the conventional configuration. Another similar sharp decline is seen for the cantilevered case at $\tilde{x} > 35$ due to another strong shock reflected from the top wall (see also Fig. 4.20b and Fig. 4.21b). Comparison of the circulation plots of the baseline unswept cantilevered configuration (A1) and the swept cantilevered configuration (A3), in Figs. 4.13a and Figs. 4.28a respectively, illustrated the steeper fluctuations for the swept case. This is due to the stronger (and less oblique) shocks generated by the swept case and consequential intersection with the upper wall further upstream (also compare unswept and swept configurations in Figs. 4.20 and 4.21). The consequences of the behaviour of the circulation in the respective flow fields of these injectors largely governs the trends in subsequent performance parameter plots.

Mixing efficiency of both swept injectors is shown in Fig. 4.28b. As expected, an ever increasing trend is observed for each injector case. Generally, superior performance is seen for the cantilevered design but the performance gains are not as dramatic as in the case of the comparison of analogous unswept designs (see Fig. 4.13b). The main explanation is two fold. First, while appreciably different values of circulation exist in the injector near-field for these configurations, vorticity levels are comparable further downstream of injection (see Fig. 4.28a, $\tilde{x} > 15$), due primarily to the dominance of the compression ramp reflected shock in the cantilevered case. Second, the intersection of the fuel jet with the lower wall brings about the situation of a significant fraction of fuel mass located in a region remote from the main vortical activity of the flow field. This shifts the mixing mechanism in this region to the inherently slower diffusion dominated regime, to wit, the mixing becomes unenhanced for this portion of the flow field.

The relative transverse fuel jet area for both injectors is shown in Fig. 4.28c. Rapid expansion of the injected fuel is noted for both configurations with much larger values for the swept cantilevered case. The subsequent contraction of the transverse hydrogen area in the case of
the cantilevered configuration is due to the dominant pressure features previously discussed. This effect is seen to be quite severe, hindering mixing performance. Increased fuel spread is noted for the cantilevered design further downstream of injection.

Decay of H₂ mass fraction and penetration trajectory (Figs. 4.28e and d) show a similar behaviour for both injectors considered. An earlier decline in maximum fuel mass fraction is discernible for the cantilevered case reflecting the greater vorticity present in the injector near-field. The detracting effects on mixing performance (discussed above) narrow the performance gap between configurations. Also of interest is the lack of inflection point for the swept cantilevered case as opposed to the baseline unswept configuration. This is observed due to the relative instability of the main vortex structures in the case of the swept injector (compare Figs. 4.6 and 4.18) and the misalignment of vortex centers and high fuel concentration regions.

Total pressure losses are presented in Fig. 4.28f. As expected, more losses are attributed to the cantilevered design in direct consequence to increased shock strength and vortex generation. The narrow gap in loss performance is commensurate with marginal increases reflected in beneficial mixing performance reflected in the other parameters.

The mixedness measure for the swept cantilevered and swept conventional injector configurations is given in Figs. 4.14c and 4.14d respectively. Mixing performance is slightly superior in the cantilevered case. The distance required to mix all pure hydrogen is decreased (compare the streamwise profile at \( Y_{H_2} = 1 \)). A more significant cascade of hydrogen mass toward lower mass fractions is also noted. The percentage of mass mixed below \( Y_{H_2} < 0.05 \) at \( x = 40 \) is 15\% for the cantilevered case while approximately 12\% for its conventional counterpart.

### 4.3.3 One-Shock Cantilevered Ramp Injector Configurations

Further insight into the characteristics of the cantilevered ramp injector, and specifically the mechanisms of axial vortex generation, are provided by Configurations B and C which prescribe zero compression and expansion angles (\( \alpha_c \) and \( \alpha_e \)) respectively. These configurations provide for one main shock formation in the flow field which serves to accentuate either cross-stream shear vorticity or baroclinic torque generated vorticity.

Figure 4.29 portrays the cantilevered configuration B1 with select hydrogen mass fraction contours. The injector geometry reveals a zero compression angle for the top surface of the injector. Evolution of the injected hydrogen fuel jet is seen to proceed in a similar manner as the baseline cantilevered case (A1). Strong entrainment of air into the jet with marked stretching of the contours in the spanwise (\( y \)) direction is noted. The increased domain height also allows substantial lift of the fuel downstream of injection (increased downstream domain height is a result of matched fuel/air inflow area ratios for all configurations). A more detailed
depiction of the injected fuel behaviour is given by Fig. 4.30. Generally, favourable qualitative mixing performance is observed, supporting the efficacy of the cantilevered injector design.

Pressure contours for Configuration B1 are given in Fig. 4.31. The absence of a shock above the injector is clear due to the zero compression angle. Also evident is the shock generated by turning the flow parallel to the lower surface at the base of the expansion trough. The height of the mixing domain facilitates only one reflection of this shock from the top surface at $\tilde{x} \approx 35$. As a result, a favourable pressure gradient exists for the generation of baroclinic torque of a consistent sense throughout the streamwise distance considered (no interaction of the reflected shock and fuel jet is noted).

These observations are also noticed in the plots of velocity vectors in Fig. 4.32. Again, the main flow feature apparent is the pair of counter-rotating vortices which remain relatively stable throughout the computational domain. While one of the design intents of Configuration B1 was to emphasize baroclinic torque vortex generation, a substantial source of vorticity is the cavity under the cantilevered injector. This further supports the contention that these vortices are the main characteristic of this injector type and are primarily responsible for improved mixing performance.

Plots of flow field parameters for Configuration C1 are given in Figs. 4.33–4.36. Markedly different hydrogen mass fraction signatures are seen for this configuration. Decreased stretching of the fuel jet in the spanwise direction is noted in comparison to Configuration B1 (see Fig. 4.29) or the baseline configuration (A1 in Fig. 4.3). This emphasizes the beneficial affects of baroclinic torque on the fuel/air mixing in the near-field of the injector as this mechanism is largely absent in this configuration. Absence of a shock in the lower portion of the pressure plots (Fig. 4.35) lends credence to the assertion of minimal baroclinic effects near the injection plane ($\tilde{x} = 0$). The flow field of this configuration however, is not devoid of vorticity generated by means of baroclinic torque. The decreased domain height results in several reflections of the shock generated by the compression surface of the ramp. Review of hydrogen mass fraction contours, pressure contours and velocity vectors (Figs. 4.34–4.36) reveals an intersection of the reflected shock with the fuel jet shortly after injection. This proves to impart vorticity of an opposite sense to that generated upstream of the injection plane by the volume under the cantilevered injector. Contraction of the fuel profile is evident in the first half of the series of streamwise locations near the top of the contours. Improved fuel jet expansion in the spanwise direction can be seen for $\tilde{x} > 20$ due, in part, to a favourable pressure gradient producing beneficial vorticity via baroclinic torque. A general dominance of vorticity of a positive sense is observed further downstream of injection. Good qualitative mixing behaviour for this configuration is noted in the far-field of the domain.
Comparison with Conventional Ramp Injector

The cantilevered ramp injectors presented in the preceding section are compared to their conventional counterparts in the present section. These configurations are designated B2 and C2 respectively.

Figure 4.37 shows the conventional ramp injector configuration with zero compression angle (Configuration B2). Evolution of the injected hydrogen is further displayed in Fig. 4.38 which gives the hydrogen mass fraction contours for this configuration. Comparison of fuel injection profiles with the corresponding cantilevered configuration (B1 in Fig. 4.30) reveals qualitatively superior mixing performance for the cantilevered design. Appreciable increase in interfacial area is noted in the near-field of the injector (compare especially plots at $\bar{x} = 10$). Improved spread and fuel dissipation (as witnessed by the lack of contour levels of high value) is apparent, particularly for increased distances downstream of injection.

Comparison of cantilevered and conventional zero compression angle injector configurations is given for a number of performance parameters in Fig. 4.39. The mixing performance increase for the cantilevered injector configuration is reflected in several plots. Vortex formation under the cantilevered injector provides for increased circulation in the range $-7 < \bar{x} < 0$ over the conventional design. While the normal decrease in circulation is realized immediately downstream of injection for Configuration B1, more remarkable is the substantially greater contribution to circulation added by baroclinic torque vortex generation over the range $2 < \bar{x} < 8$. This performance advantage is maintained further downstream such that levels of circulation are consistently higher for the cantilevered case over the conventional injector. Gains in circulation are realized as mixing performance gains as measured by the mixing efficiency and relative fuel jet area (Figs. 4.39 b and c respectively). Both measures show higher values over the range considered for the cantilevered design culminating in a 19\% increase at $\bar{x} = 40$. A dramatic difference in decay of maximum hydrogen mass fraction as a function of downstream distance is noted in Fig. 4.39e. Earlier and more complete dissipation of higher mass fractions for the cantilevered injector is seen. At the downstream end of the computational domain the maximum mass fraction for Configuration B1 (cantilevered) is 0.35 compared to a value of 0.66 for Configuration B2 (conventional). Gains in mixing performance come at the cost of a 7\% increase in total pressure losses as shown in Fig. 4.39f. The disproportional performance gain versus increased flow losses imply loss-effective mixing augmentation of the cantilevered design over the conventional ramp injector for this configuration.

Figure 4.40 shows the conventional ramp injector configuration with zero expansion angle (Configuration C2). Hydrogen mass fraction contours for this injector are given in Fig. 4.41. Comparison of hydrogen mass fractions with the zero expansion angle cantilevered configu-
ration (C1 in Fig. 4.34) show very different behaviour for the injected fuel. The conventional configuration (C2) shows the dominance of cross-stream shear generated vorticity in the injector near-field. Indeed, due to the lack of vortex generation under the injector (as is the case for the cantilevered design) this is the only major contribution to large-scale vortex structures. The fact that these vortices are generated near the top of the injected fuel profile creates a distinct waist in the contours at \( \bar{x} < 10 \) which leaves a significant portion of the injected fuel remote from vortex activity. Since vortices are not created from below due to baroclinic torque in the near-field for this configuration, no lift of the lower portion of the fuel is provided. This situation persists over the downstream range considered, resulting in constant contact of the fuel with the lower wall and a large region of unmixed fuel along the lower portion of the injector symmetry plane. The remoteness of the vortex activity in this flow field from a significant portion of the fuel sets the stage for inferior mixing performance compared to the cantilevered design. This is qualitatively reflected in the fuel mass fraction profiles and also in a comparison of performance parameters between these configurations.

Comparison of several performance parameters for the zero expansion angle configurations (C1 and C2 for cantilevered and conventional injectors respectively) is furnished by Fig. 4.42. Significantly higher levels of circulation are present in the flow field of the cantilevered case over the conventional case. This underscores the efficacy of the vortices generated by the cavity under the cantilevered fuel injector. The mixing efficiency plot displays consistently higher values for the cantilevered configuration with an increase of 27% at a streamwise location of \( \bar{x} = 40 \). Similarly superior performance is noted in plots of relative fuel area, and fuel penetration trajectory. The large unmixed portion of fuel near the lower wall in the case of the conventional configuration is primarily responsible for only slight maximum fuel mass fraction decay. This is contrasted with a very favourable decay in the case of the cantilevered injector. For this particular injector, an increase in flow losses of 40% is denoted for the cantilevered case over the conventional configuration in Fig. 4.42f. Increased projected area of the ramp top surface for the cantilevered case produces stronger shocks. Several reflections of this stronger shock are mainly responsible for greater flow losses. Further, these shock structures do not serve to augment mixing in many cases due to their distance from the fuel/air interface (compare plots at \( \bar{x} = 40 \) for Figs. 4.34 and 4.35 for example).

4.3.4 Extended Cantilevered Ramp Injector Configuration

A design feature of the cantilevered ramp injector is that the injection plane must not be coincident with the “baroclinic shock” (the shock anchored at the base of the expansion trough and primarily responsible for baroclinic axial vortex generation). Extension of the cantilevered
portion of the injector provides for earlier deposition of baroclinic torque and holds the potential for improved mixing performance over the baseline design. Fig. 4.43 shows the extended cantilevered ramp configuration (E1) with hydrogen mass fraction contours. It is clear that the base of the expansion trough is upstream of the injection plane (\(\bar{x} \approx -2\) and \(\bar{x} = 0\), respectively).

Flowfield visualization for this configuration is provided by Figs. 4.44 – 4.46 displaying hydrogen mass fraction contours, pressure contours and velocity vectors. Excellent qualitative mixing performance is evident. Early and rapid bulk mixing of the fuel and air is shown through air entrainment from under the fuel jet. Clear distortion and stretching of hydrogen mass fraction contours is shown. Good diffusion and fuel lift, or penetration, is witnessed in planes remote from the injector. Pressure contours show a contiguous baroclinic shock in the spanwise direction in the lower portion of the near-field plots. Velocity vector plots (Fig. 4.46) show the main vortex activity in the flow field. The signature pair of large scale counter-rotating vortices is realized. Careful review of the plot at \(\bar{x} = 0.1\) reveals early formation of the shock structure at the very bottom of the plot. Further, the interference of the expanding hydrogen jet with the vortex pair under the injector is qualitatively moderate. The progress and influence on circulation of the baroclinic shock is clearly demonstrated in the first half of the plot series.

**Comparison with Baseline Injector**

Measures of mixing performance comparing the extended cantilevered ramp injector (Configuration E1) to the baseline cantilevered injector (Configuration A1) is show in Fig. 4.47. The circulation in the transverse planes displays similar evolution for both configurations with a slightly higher peak reached for the extended injector. This can be attributed to the slight increase in injector length for this configuration (the distance below the cantilever at the injection plane, \(\Delta z\), is matched which dictates a different overall injector length). Most significant is the smaller decrease in circulation for the extended cantilevered case (E1) just downstream of injection. The velocity and pressure distribution of the air at the injection plane provide more favourable fuel injection conditions. The pressure under the injector is slightly increased which, in turn, decreases the expansion of the injected fuel jet and thus, less interference with the vortices generated in the cavity under the injector is realized. Coupled with this, increased injector length and earlier shock formation under the injector provide for more vigorous vortices with a more parallel streamwise trajectory which mitigate fuel expansion and wall interaction effects. Comparison of Figs. 4.44 and 4.46 show stronger vortices and earlier shock formation in the planes \(\bar{x} < 6\). The result of this is greater circulation in the flow field downstream of injection which may be utilized for mixing augmentation. This is reflected in the plot of mixing efficiency with shows a marginal yet significant performance advantage for the
extended design (approximately 7%). Other mixing performance plots reveal similar trends for both configurations. Significantly, the total pressure loss is very comparable for both configurations. The result of increased mixing performance with the absence of appreciably increased losses is a promising one. Moreover, performance results indicate that optimization of the cantilevered design is possible to further increase the mixing performance of this new design.

4.3.5 Opposing Wall Cantilevered Ramp Injector Configuration

The baseline cantilevered ramp injector (A1) was investigated in a configuration which entails injector arrays mounted on opposing walls (upper and lower) of a duct, staggered in the spanwise direction. This configuration is designated Configuration D1. The distance between symmetry planes of spanwise adjacent injectors (mounted on opposite walls) is 2.4 reference lengths while the downstream domain height is 5.7 reference lengths, resulting in a fuel/air equivalence ratio of $\phi = 1.0$. All other physical injector geometry parameters are shared with the baseline configuration A1 (see Table 4.1).

Flow visualization for this configuration is provided by hydrogen mass fraction contours, static pressure contours and velocity vectors at select transverse planes downstream of injection given in Figs. 4.48 - 4.50. Data has been mirrored in these plots about the symmetry plane $\bar{y} = 0$. Assistance in interpreting these figures is provided by Fig. 4.51 which shows the pressure fields at two $xz$ planes. The injectors on the upper walls are outlined near the top of each plot.

The hydrogen signatures for several downstream planes are seen in Fig. 4.48. Focusing on the injected hydrogen jet in the lower portion of the plots, a similar metamorphosis of the fuel as in the baseline (A1) case is observed for this configuration in the injector near-field ($\bar{x} < 10$). Specifically, evidence of a strong pair of counter-rotating vortices under the injected fuel is implied, which envelopes the air from below, given rise to characteristic fuel profiles. Further downstream however, the characteristic features of the flow change. Interaction of the flow structures generated by opposing injectors are largely responsible. Excellent penetration of the fuel is observed such that, at the end of the streamwise domain, the $H_2$ occupies a large central region of the duct, clear from wall boundaries. This behaviour is highly desirable for combustion.

Plots of static pressure (Figs. 4.49 and 4.51) reveal a complex portrait. The fuel injected from a given single injector is subject to the shocks and pressure fields it produces (mainly the compression surface shock generating cross-stream shear vorticity and the shock at the base of the expansion trough generating baroclinic torque) which promote improved fuel/air mixing, as well as pressure structures generated by injectors on the opposing walls which, in the main, detract from complimentary vortex generation. These reflected shocks can be quite strong due
to coupling of a reflected shock and one generated by a ramp injector on the opposing wall (see Fig. 4.51). The shocks created on the ramp compression surface are clearly visible in the plots. After injection, these shocks merge into a high pressure corrugated band separating individual injectors. Evidence of this is especially clear in the plots at \( \xi = 4 \) and \( \xi = 6 \) which show two thin, oblong, white (high pressure) regions focused near the plot centers. This high pressure region serves to retard fuel penetration in the near-field (also, see evidence of low penetration in the fuel mass fraction contours in Fig. 4.48, \( \epsilon < 20 \)). Further downstream the pressure field reduces to a more uniform distribution. Fig. 4.51 also serves to display the intersection of fuel/air boundaries (density gradients) with pressure gradients in a given \( xz \) plane. The fuel from a given injector unit is seen to intersect two main shocks in this plot. The first is just downstream of injection. This shock, as previously discussed, is responsible for a large baroclinic torque contribution to vortex generation in the injector near-field and has a favourable impact on mixing enhancement. The second major intersection is witnessed at a downstream location of approximately \( 25 < \xi < 30 \). This shock produces vortices, via baroclinic torque, of opposite sense to upstream generation. The effect on mixing performance is not immediately clear. While it is reasonable to assume that vortex generation of opposite sense is deleterious to mixing performance, due to decreased complimentary circulation, this vorticity can provide a destabilizing effect on the established vortices which, in turn, promote a more random distribution of transverse velocities which have the potential to improve mixing. Some overall mixing performance parameters will be discussed shortly.

Fig. 4.50 provides velocity vectors in the corresponding transverse planes. Again, similar general vortex evolution as the baseline case (A1) is observed for a single injector unit in the injector near-field. The strong vortex pair, characteristic of cantilevered injectors, is evident immediately downstream of injection. Further downstream, these vortex pairs are disrupted by the complex pressure field (especially at \( \xi = 25 \) where two shocks intersect) only to recover to organized structures which migrate toward the centre of the duct. Interaction of vortices generated from opposing injectors in seen in the far-field. Comparison of the fuel mass fraction plots and the velocity plots (Figs. 4.48 and 4.50 respectively) reveal good qualitative mixing performance in the regions between vortices. Stretching and broadening of gradients of the mass fraction contours is seen.

Performance parameters for Configuration D1 are displayed in Fig. 4.52. A large variation in circulation is seen in Fig. 4.52a. The initial sharp increase followed by a decrease immediately downstream of injection is consistent with the behaviour of the baseline configuration. Similarly, the rise in circulation in the range \( 2 < \xi < 10 \) parallels the trends noted for Configuration A1. A difference in duct height and, moreover, vastly different pressure fields, produce
a departure in circulation measure downstream of $\bar{x} \approx 10$. The large decrease in circulation at $\bar{x} > 25$ can be attributed to adverse baroclinic torque vortex generation (see pressure plots above) combined with interaction of downstream vortices which produce vorticity of opposite sense.

Favourable mixing performance is reflected in the plots of mixing efficiency, relative transverse fuel area and decay of maximum $H_2$ mass fraction. A continuously increasing trend in mixing efficiency is seen with values very similar to the baseline configuration. A large spread in the fuel jet area is seen, growing to approximately six times its original area at $\bar{x} = 40$. Steady decay of maximum fuel mass fraction is seen downstream of $\bar{x} \approx 10$.

The total pressure losses associated with the opposing wall injector configuration (D1) is shown in Fig. 4.52e. Slightly more total pressure loss is realized for the opposing wall configuration (especially in the injector near-field) compared to the baseline configuration (A1) with similar trends observed.

### 4.3.6 Incident Boundary Layer Study

It is of interest to observe the affect on mixing performance of an upstream boundary layer profile incident upon a cantilevered ramp fuel injector. A five meter long two-dimensional duct section was annexed to the computational domain upstream of the primary injector domain (see discussion in Sec. 2.2.3 and Fig. 2.4) for Configuration A1. Thus, the freestream flow developed thick boundary layer profiles which were used as inflow to the injector flow field. These boundary layer profiles are given in Fig. 4.53 and represent the variation of streamwise velocity and temperature in the $z$ direction (constant in the spanwise, $y$, direction). A boundary layer of approximately 0.4 reference lengths is seen (note that in this case the height and width of the fuel injection plane is one reference length).

The incident boundary layer displaces momentum near the injector surfaces which serve to alter the effective geometry seen by the flow. This is demonstrated in Fig. 4.54 which displays the pressure contours with the stoichiometric fuel mass fraction contour for the injector symmetry plane ($\bar{y} = 0$) and a plane at the spanwise extreme of the injector ($\bar{y} = 0.5$). A broadening of the pressure gradient on the top compression surface of the ramp is seen (compare with Figs. 4.20 and 4.21). The sharp change in geometry due to the start of the compression ramp is, in effect, smoothed to a more rounded transition by the boundary layer which produces more of a compression fan than a shock on the top surface of the ramp. As the freestream flow is expanded into the troughs between the injectors, the boundary layer thickness is increased. At the base of the expansion troughs ($\bar{x} = 0$) the freestream flow is turned parallel to the lower wall. The thick boundary layer causes a similar momentum displacement effect and the impact...
of the sharp geometry change on the flow is mitigated. A similar broadening of the pressure gradient is realized for this shock. Fig. 4.54 shows a generally weaker baroclinic shock when compared to the baseline injector configuration (A1).

An interesting result of this configuration is the reduced expansion of the injected fuel into the surrounding air immediately after injection. This can be seen by both the pressure contours and $H_2$ contour shown in Fig. 4.54 compared to Figs. 4.20 and 4.21. The boundary layer profile generated by the upstream duct is accompanied by an increased air pressure seen by the injector domain due to the attendant oblique shock train generated by the boundary layers. This, in concert with the decreased momentum in the expansion troughs and the resulting pressure increase, provides higher air pressures under the cantilevered injector. A decrease in the pressure gradient between the fuel and air under the injector results, and consequently, reduced fuel expansion. This mechanism has some unforeseen mixing performance implications (to be discussed below). Also of interest in Fig. 4.54 is the general trajectory of the fuel jet. It is noted that, in the main, the fuel is kept remote from the boundary layer regions near the lower (and upper) walls.

Evolution of the injected fuel jet for this configuration is shown in Fig. 4.55. Plots very similar to the baseline configuration (see Fig. 4.4) are shown. Curiously, slightly improved entrainment of the air into the hydrogen fuel jet is realized in the near-field plots ($\bar{x} < 4$) for the case with an incident boundary layer (BL-A1). This is due to the decreased expansion of the fuel jet and consequential reduced interference with the vortices created in the cavity under the cantilevered injector. Remarkably, these vortices, as evidenced by the deformation of hydrogen mass fraction contours, are seen to remain a dominant flow field feature despite the incident boundary layer. Overall, the qualitative behaviour of the fuel jet dynamics is seen to have been affected little by the introduction of an incident boundary layer. These observations lend credibility to the cantilevered ramp injector design and prove the efficacy of the design in potential practical implementations.

The mixing performance of this injector configuration is elucidated further via comparison of performance parameters to the baseline configuration; see Fig. 4.56. Circulation upstream of fuel injection is similar for both configurations with slightly higher values for the baseline configuration. Just downstream of fuel injection however, a significant difference is noted. The sharp decline in circulation has been largely curtailed for the configuration with an incident boundary layer (BL-A1). This provides for consistently greater circulation of this configuration downstream. As previously mentioned, decreased jet expansion toward the lower wall and less detrimental interaction of the fuel jet and cavity generated vortices accounts for this behaviour. This unexpected result points toward design improvement by increasing the pres-
sure under the cantilevered injector. A steeper decline in circulation in the region $12 < r < 25$
for Configuration BL-A1 is due to slightly stronger reflected shock structures as a result of
increased freestream air pressure. (see Fig. 4.54 and Figs. 4.20 and 4.21).

Increased circulation of Configuration BL-A1 does not translate to an increase in mixing
performance as seen in Fig. 4.56b. However, slightly increased mixing performance is shown
by the relative fuel area and maximum fuel mass fraction decay in plots Fig. 4.56c and e.

Similar total pressure loss trends are shown for the two configurations considered with an
initial offset for Configuration BL-A1 due to the losses incurred by the flow in the upstream
duct section.

Overall, mixing performance measures are seen to be relatively independent of the incident
boundary layer used for the limited comparison.

4.3.7 Far-Field Study

A study was undertaken to understand the mixing characteristics of the injected hydrogen jet far
downstream of the injection plane. To facilitate this, an additional downstream computational
domain was added and solved comprising the streamwise domain $40 < r < 70$ (sub-domain $SD_3$; see Section 2.2.3). In the present section this downstream domain is denoted as “far-field”. The far-field domain of four injector configurations were investigated, designated as FF-A1, FF-A2, FF-A3 and FF-A4, corresponding to Configurations A1–A4 respectively (see Table 4.1).

The dynamics of the hydrogen jet in the far-field are seen in Fig. 4.57 which displays
mass fraction contours for the baseline cantilevered configuration, A1. The general structure
of the fuel jet is maintained over the far-field domain: two nodes of high concentration fuel
mirrored about the injector symmetry plane at $\hat{y} = 0$. Maintenance and stability of the main
pair of counter-rotating vortices can be inferred from these dynamics (plots of velocity vectors
confirm this; not shown). A migration of the two main fuel nodes (centered around the highest
mass fraction contour) toward the top of the domain is noted. In the spanwise direction the
nodes initially move toward the injector symmetry plane (i.e. closer together) followed by a
spanwise separation further downstream ($r > 64$). Consideration of jet dynamics over the entire
domain, leads to a conclusion of an oscillatory migration of the jet in the spanwise direction
consisting of alternate expansion and compression (nodes moving together then apart). A
similar behaviour is noted in the $z$ direction. Initially, the bulk of the fuel is contained within
a relatively narrow band in the $z$ direction. A stretching of the high mass fraction contours
in the $z$ direction over the approximate domain $46 < r < 64$ followed by a contraction further
downstream in seen. Pressure gradients of opposite sense resulting in alternating baroclinic
torque deposition plays a role in governing the fuel jet dynamics. This fuel jet deformation has beneficial mixing implications as noted by the decrease of maximum contour level from 0.5 to 0.3 over the streamwise domain considered. Broadening of the hydrogen gradients, implying more uniform fuel distribution, is evident over the far-field domain.

Performance parameters for the aforementioned configurations are shown in Fig. 4.58 for the entire streamwise domain considered, including the far-field domain. This figure also serves to facilitate comparison of mixing efficacy between cantilevered ramp injectors (A1: unswept, A3: swept) and conventional ramp injectors (A2: unswept, A4: swept). Overall, trends in the far-field domains are maintained from upstream domains.

The circulation of the configurations considered decline in the first portion of the far-field domain followed by recovery in the latter portion. The influence of the shock structures previously outlined account for this decrease. An additional reflection of shocks serves to increase circulation in the later portion of the far-field domain ($ \bar{x} > 55$).

Trends in mixing efficiency (Fig. 4.58b) denote somewhat decreased mixing performance of the injectors in the far-field domain. This is to be expected as the strength of the mixing augmentation mechanisms decrease with increasing downstream distance. Further, existing vorticity is degraded via viscous forces as the flow progresses downstream. Notable is the superior performance of the swept cantilevered configuration (A3).

The expansion and contraction of the fuel jet mentioned previously is reflected in the curves of relative fuel jet area in Fig. 4.58c. An oscillatory behaviour of the hydrogen area is seen for the far-field domain ($40 < \bar{x} < 70$) with a generally increasing trend implying beneficial mixing performance over this downstream range.

Fig. 4.58d displays the fuel jet penetration for the configurations considered. A leveling of jet lift is seen, reflecting the finite domain height imposed by the top wall of the mixing duct.

A similar leveling of the decay of maximum fuel mass fraction is observed in Fig. 4.58e for the far-field domain. Decreased intensity of vortex activity and alignment of vortex and fuel mass fraction centers is responsible.

Continually decreasing values for the total pressure loss parameter is evident in Fig. 4.58f. The decrease in vortex activity and shock strength is manifest in less severe total pressure losses over the far-field streamwise domain.
Figure 4.2: Hydrogen mass fraction contours at select transverse planes for unswept cantilevered ramp injector (Configuration A1), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
Figure 4.3: Hydrogen mass fraction contours at select transverse planes for unswept cantilevered ramp injector (Configuration A1), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
**FIGURE 4.4:** Hydrogen mass fraction contours for selected transverse planes for unswept cantilevered ramp injector (Configuration A1). Contour range: $0.05 < \phi_{H_2} < 1.0$, interval: 0.05.
Figure 4.5: Pressure contours for selected transverse planes for unswept cantilevered ramp injector (Configuration A1). Contour range: $0 < P < 30000$ Pa, interval: $1000$ Pa.
FIGURE 4.6: Velocity vectors for selected transverse planes for unswept cantilevered ramp injector (Configuration A1).
Figure 4.7: Hydrogen mass fraction carpet plots for selected transverse planes for unswept cantilevered ramp injector (Configuration A1).
FIGURE 4.8: Hydrogen mass fraction contours at select transverse planes for unswept conventional ramp injector (Configuration A2), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
FIGURE 4.9: Hydrogen mass fraction contours for selected transverse planes for unswept conventional ramp injector (Configuration A2). Contour range: $0.05 < \chi_{H_2} < 1.0$, interval: 0.05.
Figure 4.11: Velocity vectors for selected transverse planes for unswept conventional ramp injector (Configuration A2).
Figure 4.12: Hydrogen mass fraction carpet plots for selected transverse planes for unswept conventional ramp injector (Configuration A2).
(a) Circulation (non-dimensionalized by $u_c \sqrt{A_f}$).

(b) Mixing efficiency.

(c) Relative transverse fuel jet area.

(d) Fuel penetration trajectory.

(e) Maximum H$_2$ mass fraction.

(f) Mass averaged total pressure.

Figure 4.13: Performance parameters as a function of downstream distance for baseline unswept cantilevered ramp injector (Configuration A1) and corresponding conventional ramp injector (Configuration A2).
Figure 4.14: Mixedness measure for: a) baseline unswept cantilevered configuration (A1); b) unswept conventional configuration (A2); c) swept cantilevered configuration (A3), and; d) swept conventional configuration (A4).
Figure 4.1: Hydrogen mass fraction contours at select transverse planes for swept cantilevered ramp injector (Configuration A3), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
Figure 4.16: Hydrogen mass fraction contours for selected transverse planes for swept cantilevered ramp injector (Configuration A3). Contour range: $0.05 < Y_{H_2} < 1.0$, interval: 0.05.
FIGURE 4.18: Velocity vectors for selected transverse planes for swept cantilevered ramp injector (Configuration A3).
FIGURE 4.19: Hydrogen mass fraction carpet plots for selected transverse planes for swept cantilevered ramp injector (Configuration A3).
CHAPTER 4. NUMERICAL INJECTOR FLOWFIELD RESULTS

(a) Unswept cantilevered ramp injector; Configuration A1.

(b) Swept cantilevered ramp injector; Configuration A3.

FIGURE 4.20: Pressure contours in $xz$ plane with stoichiometric $H_2$ mass fraction contour superimposed for cantilevered ramp injectors; $\bar{y} = 0$.

(a) Unswept cantilevered ramp injector; Configuration A1.

(b) Swept cantilevered ramp injector; Configuration A3.

FIGURE 4.21: Pressure contours in $xz$ plane with stoichiometric $H_2$ mass fraction contour superimposed for cantilevered ramp injectors; $\bar{y} = 0.5$. 
Figure 4.22: Velocity vectors at transverse injection plane ($\tilde{x} = 0$) for Configurations A1, A2, A3, A4.
Figure 4.23: Hydrogen mass fraction contours at select transverse planes for swept conventional ramp injector (Configuration A4), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
FIGURE 4.24: Hydrogen mass fraction contours for selected transverse planes for swept conventional ramp injector (Configuration A4). Contour range: $0.05 < Y_{H_2} < 1.0$. Interval: 0.05.
FIGURE 4.26: Velocity vectors for selected transverse planes for swept conventional ramp injector (Configuration A4).
FIGURE 4.27: Hydrogen mass fraction carpet plots for selected transverse planes for swept conventional ramp injector (Configuration A4).
(a) Circulation (non-dimensionalized by $u_* \sqrt{A_f}$).

(b) Mixing efficiency.

(c) Relative transverse fuel jet area.

(d) Fuel penetration trajectory.

(e) Maximum H$_2$ mass fraction.

(f) Mass averaged total pressure.

Figure 4.28: Performance parameters as a function of downstream distance for swept cantilevered ramp injector (Configuration A3) and corresponding swept conventional ramp injector (Configuration A4).
Figure 4.29: Hydrogen mass fraction contours at select transverse planes for unswept cantilevered ramp injector (Configuration B1), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
Figure 4.30: Hydrogen mass fraction contours for selected transverse planes for unswept cantilevered ramp injector (Configuration B1). Contour range: 0.05 < \( \gamma_{H_2} \) < 1.0, interval: 0.05.
Figure 4.31: Pressure contours for selected transverse planes for unswept cantilevered ramp injector (Configuration B1). Contour range: $0 < P < 30000$ Pa, interval: 1000 Pa.
Figure 4.32: Velocity vectors for selected transverse planes for unswept cantilevered ramp injector (Configuration B1).
Figure 4.33: Hydrogen mass fraction contours at select transverse planes for unswept cantilevered ramp injector (Configuration C1), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
Figure 4.34: Hydrogen mass fraction contours for selected transverse planes for unswept cantilevered ramp injector (Configuration C1). Contour range: $0.05 < Y_{H_2} < 1.0$. Interval: 0.05.
FIGURE 4.35: Pressure contours for selected transverse planes for unswept cantilevered ramp injector (Configuration C1). Contour range: $0 < P < 30000$ Pa, interval: 1000 Pa.
Figure 4.36: Velocity vectors for selected transverse planes for unswept cantilevered ramp injector (Configuration C1).
Figure 4.37: Hydrogen mass fraction contours at select transverse planes for unswept conventional ramp injector (Configuration B2), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
Figure 4.38: Hydrogen mass fraction contours for selected transverse planes for unswept conventional ramp injector (Configuration B2). Contour range: $0.05 < Y_{H_2} < 1.0$, interval: 0.05.
Figure 4.39: Performance parameters as a function of downstream distance for cantilevered ramp injector (Configuration B1) and corresponding conventional ramp injector (Configuration B2).
Figure 4.40: Hydrogen mass fraction contours at select transverse planes for unswept conventional ramp injector (Configuration C2), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
FIGURE 4.4.1: Hydrogen mass fraction contours for selected transverse planes for unswept conventional ramp injector (Configuration C2). Contour range: $0.05 < H_2 < 1.0$. Interval: 0.05.
(a) Circulation (non-dimensionalized by $u_\infty \sqrt{A_f}$).
(b) Mixing efficiency.
(c) Relative transverse fuel jet area.
(d) Fuel penetration trajectory.
(e) Maximum H$_2$ mass fraction.
(f) Mass averaged total pressure.

**Figure 4.42**: Performance parameters as a function of downstream distance for cantilevered ramp injector (Configuration C1) and corresponding conventional ramp injector (Configuration C2).
FIGURE 4.43: Hydrogen mass fraction contours at select transverse planes for unswept cantilevered ramp injector (Configuration E1), with shaded injector geometry surfaces (data mirrored around injector symmetry plane).
FIGURE 4.44: Hydrogen mass fraction contours for selected transverse planes for unswept extended cantilevered ramp injector (Configuration E1). Contour range: 0.05 < $Y_{Hz}$ < 1.0, interval: 0.05.
FIGURE 4.45: Pressure contours for selected transverse planes for unswept extended cantilevered ramp injector (Configuration E1). Contour range: \(0 < P < 30000\) Pa, interval: 1000 Pa.
Figure 4.46: Velocity vectors for selected transverse planes for unswept extended cantilevered ramp injector (Configuration E1).
(a) Circulation (non-dimensionalized by $u_{\infty}\sqrt{A_f}$).

(b) Mixing efficiency.

(c) Relative transverse fuel jet area.

(d) Fuel penetration trajectory.

(e) Maximum H$_2$ mass fraction.

(f) Mass averaged total pressure.

**Figure 4.47**: Performance parameters as a function of downstream distance for baseline unswept cantilevered ramp injector (Configuration A1) and extended unswept cantilevered ramp injector (Configuration E1).
FIGURE 4.48: Hydrogen mass fraction contours for selected transverse planes for unswept cantilevered ramp injector (Configuration D1). Contour range: $0.05 < Y_{H_2} < 1.0$, interval: 0.05.
Figure 4.50: Velocity vectors for selected transverse planes for unswept cantilevered ramp injector (Configuration D1).
Figure 4.51: Pressure contours in $xz$ plane with stoichiometric H$_2$ mass fraction contour superimposed for opposing wall cantilevered injector; Configuration D1.
Figure 4.52: Performance parameters as a function of downstream distance for unswept cantilevered ramp injector (Configuration D1).

(a) Circulation (non-dimensionalized by $u_x \sqrt{A_f}$).

(b) Mixing efficiency.

(c) Relative transverse fuel jet area.

(d) Maximum $H_2$ mass fraction.

(e) Mass averaged total pressure.
Figure 4.53: Incident boundary layer profiles immediately upstream of injector.

Figure 4.54: Pressure contours in $xz$ plane with stoichiometric $H_2$ mass fraction contour superimposed for cantilevered injector with incident boundary layer; Configuration BL-A1.
Figure 4.55: Hydrogen mass fraction contours for selected transverse planes for unswept cantilevered ramp injector with incident boundary layer (Configuration BL-A1). Contour range: $0.05 < Y_{H_2} < 1.0$, interval: 0.05.
FiguRE 4.56: Performance parameters as a function of downstream distance for baseline unswept cantilevered ramp injector (Configuration A1) and baseline unswept cantilevered ramp injector with incident boundary layer (Configuration BL-A1).
Figure 4.57: Hydrogen mass fraction contours for selected far-field transverse planes for unswept cantilevered ramp injector (Configuration A1). Contour range: $0.05 < \phi_H < 1.0$, interval: 0.05.
(a) Circulation (non-dimensionalized by $u_\infty \sqrt{A_f}$).

(b) Mixing efficiency.

(c) Relative transverse fuel jet area.

(d) Fuel penetration trajectory.

(e) Maximum $H_2$ mass fraction.

(f) Mass averaged total pressure.

**Figure 4.58:** Performance parameters as a function of downstream distance for baseline unswept cantilevered ramp injector (Configuration A1), unswept conventional ramp injector (Configuration A2), swept cantilevered injector (Configuration A3), and swept conventional injector (Configuration A4).
Chapter 5
CONCLUDING DISCUSSION

5.1 Summary and Conclusions

Motivated by a need for more efficient propulsion devices in the hypersonic regime, contemporary research efforts have given great heed to air-breathing concepts. Air-breathing propulsion offers the substantial advantage of utilizing the oxygen available in the surrounding air rather than tankage of onboard oxidizer (as with rocket based propulsion). The most significant technological problem for implementation of hypersonic air-breathing propulsion is the process of mixing the injected fuel with the freestream oxidant. A particularly difficult challenge is presented for combination of fuel and air at very high Mach numbers and high enthalpy since combustion proceeds at supersonic velocities. Extremely short combustor residence times coupled with the requirement of limited combustor length necessitates some form of mixing augmentation over that provided by streamwise shear alone.

The present study has presented a new ramp injector design denoted as the cantilevered ramp injector, to address the need for enhanced hypervelocity mixing of hydrogen fuel with freestream air. This novel injector was developed at the University of Toronto Institute for Aerospace Studies (UTIAS) and this thesis represents its inaugural study. The cantilevered ramp injector embodies characteristics of existing conventional ramp injectors and low angled wall injection to produce increased axial vorticity which, in turn, enhance bulk fuel/air mixing.

A numerical investigation was undertaken to elucidate the salient flow field features of the cantilevered ramp injector design and to evaluate its performance in the context of mixing
efficacy and loss production compared with existing designs. A parametric study of various injector geometries and configurations were numerically solved and analyzed. Simulation of the flow field was facilitated by the construction of a three-dimensional CFD solver which modeled the laminar Navier-Stokes equations for a multi-species gas in generalized coordinates.

The analysis of the numerical solutions of the fuel injector configurations investigated herein provide the following conclusions.

It may be stated with some confidence that the cantilevered ramp injector concept has been shown to be a viable fuel delivery system for hypersonic propulsion when compared to existing designs. Significant gains in mixing performance measures have been realized by the baseline cantilevered injector over its conventional counterpart. Further, two major design objectives of the cantilevered ramp injector concept have been proven effective: 1) generation of a strong vortex pair in the cavity under the injector, and; 2) earlier and more substantial deposition of baroclinic torque (primarily due to the increased shock strength under the injected fuel jet). Both of these features have been proven to augment mixing performance and to produce additional vorticity in the cantilevered ramp case.

Insight into the structure of the cantilevered ramp injector flow field has been provided. The main distinguishing feature of a cantilevered flow field is the signature pair of counter-rotating vortices generated in the cavity under the injector. These vortices have been shown to be of significant strength and size and to beneficially contribute to fuel/air mixing performance. The vorticity generated under the fuel injector is created in the same sense as that created by baroclinic torque and cross-stream shear in the injector near-field.

Introduction of side wall sweep in the case of the present cantilevered ramp configuration provides for only marginal mixing performance gains over the unswept baseline configuration. The present study illustrated the potentially deleterious affect of strong shocks reflected from the top domain surface. Shocks reflected from the top wall impart a pressure gradient of opposite direction to that imposed by the unreflected shock. When both portions of this shock interact with the density gradient at the fuel jet/air interface, vorticity of opposite sense is deposited due to baroclinic torque. This serves to weaken the primary vortex structures. Indeed, the influence of the duct walls on the flow features and thereby the dynamics of the injected fuel jet is very appreciable. This implies the importance of judicious duct design for efficient mixing.
5.2 Suggestions for Further Research

The numerical investigation of the present study of cantilevered ramp injectors was largely aimed at a preliminary evaluation of the axial vortex generating mechanisms of a new fuel delivery system. Future avenues of research which build upon the current investigation could take the form of numerical model development and/or further parametric studies.

Additional physical phenomena could be incorporated into the present numerical model in an effort to determine mixing performance to increasing accuracy. Two of the most relevant physical phenomena to further research of the cantilevered ramp injector are turbulence modeling and combustion modeling.

Inclusion of a turbulence model will serve to further illuminate the mixing characteristics of a given injector in regions farther downstream of injection. Mixing of fuel and air is governed by increasingly smaller scale mixing with increasing downstream distance from injection. Turbulent effects become more pronounced at these scales. More realistic quantitative mixing performance is afforded by accurate modeling of turbulence. Turbulence modeling would also provide the means to investigate injector configurations with large differences in freestream and injected streamwise velocities, and thus high streamwise shear.

The ultimate goal of any fuel/air mixing effort for propulsion applications is an increase in vehicle thrust. Potentially increased thrust can be inferred from increased mixing performance but the precise gain in combustion due to mixing gains (measured in the absence of combustion) constitutes an assumption. The coupled phenomena of mixing and burning can only be numerically modeled with the inclusion of a finite-rate chemical kinetic mechanism. Inclusion of a chemical reaction model would facilitate the direct calculation of combustion and thrust gains. A complete air-breathing propulsion device utilizing cantilevered ramp injectors could be solved numerically to ascertain relevant propulsion performance parameters and compared with alternative fuel injection techniques to gauge the installed performance of injection systems. Reacting simulations would also provide meaningful insight into the flammability limits of injected fuel. The downstream location where the chemical reactions of combustion are initiated is of key concern to propulsion concepts which premix the fuel and air prior to combustion. A reacting CFD code may be employed to shed light on such practical questions.

A wide spectrum of possibilities exist for further parametric study of the cantilevered ramp fuel injector. Parametric investigations could take the form of a variation of flow conditions and/or injector geometry configurations.

One of the relevant outstanding questions is the mixing performance of an injector over a range of freestream flow conditions. Of interest is the mixing performance of a given injector
configuration as the freestream conditions change, for example, over an accelerating flight trajectory such as would be encountered by a shock-induced combustion ramjet. More basic investigations may also be carried out which evaluate the influence of injectant to freestream pressure and velocity ratios on mixing performance.

A host of geometric variations of the cantilevered ramp injector are available for study. Variation of compression, expansion and sweep angle could provide for a parametric study to elucidate optimal values for given flow conditions. It is recommended that future configurations employ a reduced compression ramp angle to mitigate shock reflection effects and reduce fuel expansion toward the lower wall. More moderate sweep angles may also be employed to reduced these effects. Further, it is conjectured that optimum values of the spanwise injector spacing, duct height, height of injector tip from the bottom wall and injector length exist for given flow conditions. These variables could also be optimized for potentially increased cantilevered ramp injector performance.

5.2.1 Alternative Injector Designs

Given the success of the cantilevered ramp injector design, several variations or departures are advanced as possible subjects for future study.

Building on the cantilevered design, an injector which retains the geometric form of the baseline cantilevered injector is proposed with the addition of a streamwise air jet in the cavity under the injector. The small base area under the injector (at the upstream extent of the cantilevered portion of the injector) may be slightly enlarged and used as an air inflow. While the practical implementation of such a scheme may be formidable, it is of interest to observe the resulting fuel jet dynamics. The addition of the extra air jet under the fuel jet would provide an extra buffer between the fuel and the high temperature region near the lower wall. Further, an increase in pressure under the injected fuel and increased streamwise momentum of the vortices created in the cantilever cavity has the potential to promote enhanced fuel penetration. Introduction of low temperature air would also provide some measure of insurance against premature fuel ignition in the injector near-field and mitigate cooling loads on the injector itself.

The cantilevered ramp injector geometries considered in the current study are comprised of flat planar surfaces. Abrupt changes in surface angle generate shocks of interest to mixing enhancement. An alternative to this design is curved surfaces which produce compression fans (in the region near the surface). A specific alteration to the cantilevered ramp design could entail gradual curved wall sections in the expansion troughs and under the cantilevered injector. This configuration would form a compression fan rather than a shock at the base of the expansion troughs and produce a distributed deposition of baroclinic torque rather than a
discrete deposition. The benefits of alternative torque deposition on the fuel/air interface is an open question and forms an interesting investigation. Further contouring of injector surfaces could take the form of various inserts located at the upstream end of the cantilever cavity. These wall contours could “guide” the freestream air into the cavity to promote enhanced vorticity generation and streamwise momentum.

The concept of side-wall sweep imposed on a ramp injector is a common method of enhancing streamwise vorticity. An alternative approach worth investigation is a negative sweep angle (such that the downstream end of the ramp is wider in the spanwise direction than the upstream end). Small negative sweep angles would produce shocks adjacent to the side walls of the injector. Reflection and interaction of these shocks would impart an addition and consistent pressure gradient on the injected fuel jet which would promote additional baroclinic torque vortex generation. In the case of the cantilevered injector, the possibility of increased cavity pressure also exists. Moderate negative sweep angles are essential to avoid excessive flow losses for this configuration.

An alternative to the conventional ramp injector design is the so-called “stunted ramp” design. This design would be very similar to the baseline conventional ramp investigated in the current study (see Fig. 2.1b) with a downstream portion of the ramp removed. This would move the fuel injection plane upstream of the base of the expansion troughs. In so doing, a “virtual cantilever” is introduced whereby the low pressure at the base of the injection plane behind the ramp would act as the cavity of the cantilevered design. Streamwise vorticity would be produced just downstream of injection. Additional streamwise vorticity is realized as the fuel jet passes through the shock at the base of the expansion troughs slightly further downstream. This shock is of increased strength in this configuration over the standard conventional configuration due to the more contiguous spanwise shock structure.
This appendix outlines various aspects which are germane to the practical implementation of the numerical code developed herein.

**A.1 Block Connectivity Schematics**

Exploded schematics of the block numbering and connectivity for various ramp fuel injector configurations are given in Figs. A.1 – A.4. The block groups in the indicated η planes are connected in the computational domain and are only separated for clarity. The lower portion of each figure gives a schematic profile view indicating the locations of the various blocks relative to the fuel injector. Planes indicated as “injector plane” correspond to the spanwise extent which contains the injector geometry while “freestream plane” indicates the volume between injectors consisting of only wall contours.

In the case of the conventional ramp injector geometry (Fig. A.1) the number of blocks in the primary sub-domain is equal to 14. For the standard cantilevered ramp injector the corresponding number of blocks is 15. Note that the only difference between the grid structure for these configurations is the extra grid block under the injector for the cantilevered ramp injector case. This is done so that grid clustering and densities can be kept consistent for both types of injectors. This provides for a more meaningful comparison of results between these two configurations as grid resolution issues are largely limited through this approach since, save for the area under the injector in the case of the cantilevered configuration, the same number
and distribution of grid cells exist in a given part of the domain for both configurations.

In the case of the extended cantilevered ramp injector configuration (Fig. A.3) 20 blocks are employed. The added number of blocks over the standard cantilevered ramp configuration is implemented to simplify individual block generation and to maintain grid resolution consistency as described above.

A schematic of an opposing wall cantilevered ramp injector configuration is given in Fig. A.4. Twenty one blocks are employed for the primary sub-domain with three planes in the $\eta$ direction. As with the other configurations, $\eta = \eta_{\text{min}}$ and $\eta = \eta_{\text{max}}$ correspond to symmetry planes.

The numbering of the blocks is significant in the current formulation since the numerical algorithm employed (specifically the temporal discretization scheme) requires inversion in a particular cell order (see Section 3.3 and Fig. 3.1). Computations are carried out for each block on $i + j + k = \text{const.}$ oblique plane sweeps. This corresponds to a forward sweep starting at the first cell at the block origin and culminating at $(i, j, k) = (i_{\text{max}}, j_{\text{max}}, k_{\text{max}})$. The blocks are numbered, and thus calculated, such as to be consistent with this approach.
Figure A.1: Block connectivity schematic for ramp injector geometry.
Figure A.2: Block connectivity schematic for cantilevered ramp injector geometry.
Figure A.3: Block connectivity schematic for extended cantilevered ramp injector geometry.
Figure A.4: Block connectivity schematic for opposing wall cantilevered ramp injector geometry.
REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


REFERENCES


