ABSTRACT

TRAVIS, JEFFREY TODD. A Computational Investigation of a Constant Volume Combustion Jet Engine. (Under the direction of Dr. William L. Roberts and Dr. Andrey V. Kuznetsov).

The constant volume combustion thermodynamic cycle, or Humphrey Cycle, if made practical in a jet engine, can provide enormous benefits over the traditional jet engine cycle, the Brayton Cycle, which is a constant pressure cycle. These advantages include increased thermal efficiency, lower specific fuel consumption, and higher specific impulse. Other just as important benefits inherent in a constant volume device include simplicity of design, the ability to miniaturize the device because of the simplicity, low cost, damage tolerance, expendability, and a high thrust-to-weight ratio. This device has the potential to provide extremely high thrust in an austere, economical package.

All of these attributes, when proven feasible, have the ability to provide a propulsion source for military and civilian applications alike, including applications in miniaturized unmanned aerial vehicles (UAVs). The purpose of this study is to computationally model and understand the physics of a possible constant volume combustion jet engine, including the combustion and fluid mechanics of such a device. Specifically, the investigation will include studying different engine geometries, valve designs and timing, various air inlet and fuel mixing schemes, operating frequencies, nozzle designs, along with the effects of a convective air stream when a supporting vehicle is in flight at various airspeeds. This study will seek to maximize thrust and minimize fuel consumption, and thus develop a jet engine with many possible future applications.
A Computational Investigation of a Constant Volume Combustion Jet Engine

by

Jeffrey Todd Travis

A dissertation submitted to the Graduate Faculty of
North Carolina State University
in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy

Aerospace Engineering

Raleigh, North Carolina

2014

APPROVED BY:

_______________________________  ______________________________
Dr. Andrey V. Kuznetsov        Dr. William L. Roberts
Committee Chair

___________________________  __________
Dr. Robert T. Nagel            Dr. Jack R. Edwards Jr.
DEDICATION

This study is dedicated to my family - my wife Anne, my daughters Holly and Chloe, and my son Niklas. Completing these studies while also working to support my family, while traveling away from home often, required their unbridled support and patience. Their years of sacrifice during this time will never be forgotten and I will spend my future years showing my gratitude.
BIOGRAPHY

The author was born in Newton, North Carolina in 1955, the son of Charles Travis and Shelby Crawford, and attended Hickory, North Carolina High School. In 1979, he received a B.S. in Mechanical Engineering from North Carolina State University and entered the U.S. Air Force as an Officer and Mechanical Engineer. After mechanical engineering assignments in Arizona and Alaska, he entered Air Force pilot training in 1982 in Arizona. His first pilot assignment was as an A-10 fighter pilot at Myrtle Beach Air Force Base, South Carolina. His next duty was as a T-38 instructor pilot at Vance Air Force Base, Oklahoma. In 1990, he became an F-16 fighter pilot, assigned to Homestead Air Force Base, Florida and King Fahd Airfield, Dhahran, Saudi Arabia. His military flying career included being a four-ship flight lead, a flight instructor, a flight commander, a functional test pilot, a standardization and evaluation pilot, and a chief of academic training. After Operation Desert Storm, the first Persian Gulf war, he elected to retire early from the Air Force, in the rank of Major. While in the Air Force, the author completed two graduate degrees – in 1987 he received a Master of Arts in Business Administration from Webster University and in 1992 he received a Master of Engineering in Mechanical Engineering from the University of Idaho. After the Air Force, the author worked for the international engineering firm, Black & Veatch, in power plant design, becoming a Professional Engineer. In 1996, he returned to flying, as an airline pilot for Federal Express. For the past 18 years, he has flown the B-727, DC-10, B-757 and is currently an A-300 captain, flying domestic and international routes. The author began his doctoral studies in aerospace engineering at North Carolina State University in 2004 under the direction of Dr. Bill Roberts and Dr. Andrey Kuznetsov.
ACKNOWLEDGMENTS

I would like to thank Dr. Bill Roberts for inviting me to join his team of researchers at the North Carolina State University Applied Energy Research Lab (AERL) in 2004 and especially for directing me toward the investigation of this constant volume combustion jet engine. The computational work in this study was perfectly suited to my life situation – fully employed, traveling often, with a busy family life – and the novel and unique nature of this study was always satisfying.

I wish to also thank my Committee members, Dr. Andrey Kuznetsov, Dr. Robert Nagel, and Dr. Jack Edwards for their time, interest and support.

Finally, I am grateful to my other AERL students, in particular, fellow student Dr. Fei Zheng, of Beijing, China. His assistance during the early portion of my computational work was vital and I will always appreciate and remember his help and friendship.
# TABLE OF CONTENTS

LIST OF TABLES........................................................................................................ix

LIST OF FIGURES.......................................................................................................x

Chapter 1 Introduction.................................................................................................1
  1.1 Constant Volume Combustion.........................................................................1
  1.2 Research Objectives.........................................................................................3
  1.3 Combustion and Fluid Mechanics Model.......................................................6
    1.3.1 Combustion Model..................................................................................6
    1.3.2 Turbulence Model...................................................................................7
  1.4 CFD Model Geometry.....................................................................................9
  1.5 Model Operation..............................................................................................15
  1.6 CFX Model.....................................................................................................16

Chapter 2 Valve Timing Optimization.................................................................23
  2.1 Valve Control Background..........................................................................23
  2.2 Valve Timing Observations..........................................................................26
  2.3 Conclusions..................................................................................................30

Chapter 3 Perimeter Inlet Geometries and Simulations...............................31
  3.1 Introduction..................................................................................................31
  3.2 Single Inlet Full Combustion Chamber......................................................31
  3.3 Dual Inlet Full Combustion Chamber..........................................................34
  3.4 Single Inlet Reduced Combustion Chamber..............................................37
  3.5 Dual Inlet Reduced Combustion Chamber................................................40
Chapter 7 Engine in a Convective Air Stream

7.1 CFX Model Setup for Engine in a Convective Air Stream

7.2 Convective Air Simulations

7.3 Conclusions

Chapter 8 Nozzle Optimization

8.1 Supersonic Nozzle Design

8.2 CFX Nozzle Optimization Simulations

8.3 Contoured Nozzle Analysis

8.4 Conclusions

Chapter 9 Thermodynamic Analysis and Thrust-to-Weight Ratios

9.1 Thermodynamic Analysis

9.2 Thrust-to-Weight Ratios

9.3 Conclusions

Chapter 10 Study Findings and Future Work

10.1 Study Findings

10.2 Future Work

REFERENCES

APPENDIX

Appendix A - Initial CFX Engine Geometry

Appendix B - CFX – Pre User Expressions for Control and Analysis

Appendix C - Typical ANSYS Geometry; Mesh; CFX-Pre, Solver, Post and Screens
Appendix D - Reactant Mixing Result Details........................................150
Appendix E - Sinusoidal Inlet and Throat Ball Valve Result Details.............151
Appendix F - 2-D Nozzle Design Program.............................................152
Appendix G - Thermodynamic Analysis Result Details............................ 159
Appendix H - Typical Valve Motor.......................................................162
Appendix I - Typical Motor Batteries...................................................164
LIST OF TABLES

Table 6-1: Simulation Results Using Relocated, Centralized Valve Sinusoidal Flow Profiles........................................................................................................................................76

Table 7-1: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 100 m/s and 200 m/s........................................................................................................................................92

Table 7-2: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 225 m/s and 250 m/s........................................................................................................................................93

Table 7-3: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 275 m/s...........................................................................................................................................94

Table 8-1: Results from 6 bar and 7 bar Nozzle Design Simulations.........................109

Table 8-2: Results from 8 bar and 10 bar Nozzle Design Simulations.......................110

Table 8-3: Simulation Results for 6, 7 and 8 bar Contoured Nozzles at 225 m/s and 275 m/s.................................................................................................................................................117

Table 9-1: Constant Volume Combustion Jet Engine Thermodynamic Analysis Summary...........................................................................................................................................128

Table 9-2: Thrust-to-Weight Ratio Summary for Constant Volume Combustion Jet Engine.................................................................................................................................................132
LIST OF FIGURES

Figure 1-1: Comparison of Humphrey Cycle and Brayton Cycle.................................2

Figure 1-2: Specific Impulse vs. Mach Number for Various Propulsion Systems..........5

Figure 1-3: Schematic of German Aeropulse used to power V1 "buzz bomb"..............10

Figure 1-4: Hobby Scale "Dyna-Jet" Pulsejet...............................................................11

Figure 1-5: Pulsejet Air Inlet Design, Courtesy of BMS Company..........................12

Figure 1-6: Initial Geometry for Constant Volume Combustion Jet Engine CFD Simulations.............................................................13

Figure 1-7: Constant Volume Combustion Jet Engine Isometric View......................14

Figure 1-8: Constant Volume Combustion Jet Engine Plan View.............................14

Figure 1-9: Constant Volume Combustion Jet Engine CFD Wedge Model...............19

Figure 1-10: Constant Volume Combustion Jet Engine and Outer Domain Wedge Model............................................................................20

Figure 1-11: Constant Volume Combustion Jet Engine Wedge Model After Meshing....21

Figure 1-12: Mesh Inflation Control Used to Decrease Node Spacing in Boundary Layer........................................................................21

Figure 2-1: Resistance Loss Coefficient for Control of Binary Inlet and Throat Valves...25

Figure 2-2: Typical Pressure Plots with Monitor Point Locations Shown..................26

Figure 2-3: Effect of Minimum Expansion Pressure on Next Cycle's Maximum Pressure.............................................................................27

Figure 2-4: Time and Time Ratios Required for Each Cycle Phase, from Trial Simulations........................................................................28

Figure 2-5: Pressure and Thrust Plots of Initial Condition Cycle and Next Full Cycle...28

Figure 2-6: Effect of Inlet Open Duration on Specific Impulse for 194hz Engine.........29
Figure 3-1: Single Inlet Full Combustion Chamber Geometry ...........................................31

Figure 3-2: Single Inlet Full Combustion Chamber Pressure, Thrust and Fuel Flow
Plots (Wedge) ...........................................................................................................32

Figure 3-3: Sequential Temperature Images of One Cycle of the Single Inlet Full
Combustion Chamber Engine Model ........................................................................33

Figure 3-4: Dual Inlet Full Combustion Chamber Geometry ...........................................34

Figure 3-5: Dual Inlet Full Combustion Chamber Pressure, Thrust and Fuel Flow
Plots (Wedge) ...........................................................................................................35

Figure 3-6: Sequential Temperature Images of One Cycle of the Dual Inlet Full
Combustion Chamber Engine Model ........................................................................36

Figure 3-7: Single Inlet Reduced Combustion Chamber Geometry .................................37

Figure 3-8: Single Inlet Reduced Combustion Chamber Pressure, Thrust and Fuel
Flow Plots (Wedge) ....................................................................................................38

Figure 3-9: Sequential Temperature Images for One Cycle of the Single Inlet Reduced
Combustion Chamber Engine Model ........................................................................39

Figure 3-10: Dual Inlet Reduced Combustion Chamber Geometry .................................40

Figure 3-11: Dual Inlet Reduced Combustion Chamber Pressure, Thrust and Fuel
Flow Plots (Wedge) ....................................................................................................41

Figure 3-12: Sequential Temperature Images of One Cycle Of the Dual Inlet Reduced
Combustion Chamber Engine Model ........................................................................42

Figure 3-13: Single Inlet Reduced Combustion Chamber and Reduced Throat
Geometry .......................................................................................................................43

Figure 3-14: Single Inlet Reduced Combustion Chamber and Reduced Throat
Pressure, Thrust and Fuel Flow Plots (Wedge) ............................................................44

Figure 3-15: Sequential Temperature Images of One Cycle of the Reduced
Combustion Chamber and Reduced Throat Engine Model .........................................45

Figure 4-1: Thrust vs. Frequency for the Single Inlet Full Combustion Chamber
Model ............................................................................................................................47
Figure 4-2: Fuel Flow vs. Frequency for the Single Inlet Full Combustion Chamber Model

Figure 4-3: Specific Impulse vs. Frequency for the Single Inlet Full Combustion Chamber Model

Figure 4-4: Thrust vs. Frequency for the Dual Inlet Full Combustion Chamber Model

Figure 4-5: Fuel Flow vs. Frequency for the Dual Inlet Full Combustion Chamber Model

Figure 4-6: Specific Impulse vs. Frequency for the Dual Inlet Full Combustion Chamber Model

Figure 4-7: Thrust vs. Frequency for the Single Inlet Reduced Combustion Chamber Model

Figure 4-8: Fuel Flow vs. Frequency for the Single Inlet Reduced Combustion Chamber Model

Figure 4-9: Specific Impulse vs. Frequency for the Single Inlet Reduced Combustion Chamber Model

Figure 4-10: Thrust vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model

Figure 4-11: Fuel Flow vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model

Figure 4-12: Specific Impulse vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model

Figure 4-13: Thrust vs. Frequency for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model

Figure 4-14: Fuel Flow vs. Frequency for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model

Figure 4-15: Specific Impulse for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model

Figure 4-16: Specific Impulse vs. Frequency for All Geometries
Figure 6-12: Sequential Images of Propane Mass Fraction During One Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves

Figure 6-13: Sequential Images of Temperature During One Maximum Thrust Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves

Figure 6-14: Sequential Images of Temperature During One Maximum Thrust Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves (continued)

Figure 6-15: Sequential Images of Temperature During One High Specific Impulse Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves

Figure 6-16: Sequential Images of Temperature During One High Specific Impulse Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves (continued)

Figure 7-1: Outer Domain Inlet Boundary Condition for Convective Air Stream

Figure 7-2: Pressure Plot for a 225 m/s and 8.56 grams/sec Convective Air Simulation. Freestream and Compressor Pressure Rises Are Shown

Figure 7-3: Temperature and Thrust Plots for a 225 m/s and 8.56 grams/sec Convective Air Simulation

Figure 7-4: Pressure and Fuel Plots for a 275 m/s and 2.28, 3.71 and 12.13 gram/sec Propane Convective Air Simulation

Figure 7-5: Convective Air at 275 m/s Sequential Temperature Images, Propane at 12.13 grams/sec

Figure 7-6: Convective Air at 275 m/s Sequential Mach Number Images, 12.13 grams/sec Propane, Expansion Phase Only, Maximum Exhaust Mach Number is 2.4

Figure 7-7: Average Engine Thrust vs. Average Engine Fuel Flow for Convective Stream of 100 m/s, 200 m/s, 225 m/s, 250 m/s and 275 m/s

Figure 7-8: Specific Impulse vs. Average Engine Fuel Flow for Convective Stream of 100 m/s, 200 m/s, 225 m/s, 250 m/s and 275 m/s

Figure 8-1: Supersonic Nozzle Flow Conditions During Expansion
Figure 8-2: Rocket Motor Blowdown Similarity to Constant Volume Combustion Jet Engine Expansion………………………………………………………………………101

Figure 8-3: Method of Characteristics for Nozzle Design………………………………………………………………………………………………………………………………103

Figure 8-4: 2-D Nozzle Design Typical MATLAB Output – 6 bar Nozzle…………………104

Figure 8-5: Validation of the 2-D Nozzle Design MATLAB Program……………………………………………………………………………………………..106

Figure 8-6: 6 bar and 7 bar Nozzle Design Geometries…………………………………107

Figure 8-7: 8 bar and 10 bar Nozzle Design Geometries………………………………108

Figure 8-8: 6, 7, 8 and 10 bar Nozzle Comparisons, Thrust vs. Fuel Flow, 225 m/s………………………………………………………………………………………………………112

Figure 8-9: 6, 7, 8 and 10 bar Nozzle Comparisons, Specific Impulse vs. Fuel Flow, 225 m/s…………………………………………………………………………………………………………113

Figure 8-10: 6, 7, 8 and 10 bar Nozzle Comparisons, Thrust vs. Fuel Flow, 275 m/s………………………………………………………………………………………………114

Figure 8-11: 6, 7, 8 and 10 bar Nozzle Comparisons, Specific Impulse vs. Fuel Flow, 275 m/s………………………………………………………………………………115

Figure 8-12: 6 bar Contoured Nozzle……………………………………………………116

Figure 8-13: 7 bar Contoured Nozzle………………………………………………116

Figure 8-14: 8 bar Contoured Nozzle…………………………………………116

Fig. 8-15: 6, 7 and 8 bar Contoured Nozzle Thrust Comparison to 6 bar Cone-Shaped Nozzle at 225 m/s…………………………………………………………………………………………118

Fig. 8-16: 6, 7 and 8 bar Contoured Nozzle Specific Impulse Comparison to 6 bar Cone-Shaped Nozzle at 225 m/s……………………………………………………………………………………119

Fig. 8-17: 6, 7 and 8 bar Contoured Nozzle Thrust Comparison to 6 bar Cone-Shaped Nozzle at 275 m/s……………………………………………………………………………………..119

Fig. 8-18: 6, 7 and 8 bar Contoured Nozzle Specific Impulse Comparison to 6 bar Cone-Shaped Nozzle at 275 m/s……………………………………………………………………120
Figure 9-1: Humphrey Cycle Pressure-Volume (P-V) and Temperature-Entropy (T-s) Diagrams

Figure 9-2: Comparison of Constant Volume Combustion Jet Engine with Other Propulsion Sources – Specific Impulse vs. Mach Number

Figure 10-1: Possible Nozzle Contour Improvements
Chapter 1  Introduction

1.1  Constant Volume Combustion

The constant volume combustion cycle, or Humphrey cycle, is similar to the Brayton cycle, except that the Brayton cycle’s constant pressure heat addition process is replaced with a constant volume heat addition process. The ideal Humphrey cycle consists of the following four processes [1]:

1. **A reversible, adiabatic (isentropic) compression of the incoming air.**
   
   Stagnation pressure and temperature increase because of the work done on the air by a compressor. Convective air from a freestream provides additional compression. Entropy is unchanged, while static pressure and density of the air increase.

2. **A constant-volume heat addition.** Heat is added from combustion, while the air is kept at a constant volume.

3. **A reversible, adiabatic (isentropic) expansion of the air.** The air is expanded, in this device’s case, through a nozzle. Stagnation temperature and pressure decrease because of the work (thrust) extracted. Entropy is unchanged, while static pressure and density of the air decrease.

4. **A constant-pressure heat rejection.** Heat is removed from the air, while the air remains at constant pressure. In this device, this occurs when the inlet opens, with expulsion of the air to the atmosphere at ambient pressure, which
is considered an infinitely large reservoir for heat storage, at constant pressure and temperature.

As seen below (Fig. 1-1) in an overlay of the Humphrey cycle on the Brayton cycle [2], the constant-volume combustion enables much higher pressures during the heat addition, which provides additional work during the expansion phase. Approximately 10% higher thermal efficiencies can be attained by this additional work, which also leads to 30–35% lower specific fuel consumption (SFC). [3]

![Humphrey Cycle vs Brayton Cycle](image)

**Humphrey Cycle**
0-1 isentropic compression
1-2 isochoric heat addition
2-3 isentropic expansion
3-0 isobaric heat rejection

**Brayton Cycle**
1-2 isentropic compression
2-3 isobaric heat addition
3-4 isentropic expansion
4-0 isobaric heat rejection

*Figure 1-1: Comparison of Humphrey Cycle and Brayton Cycle*
The challenges of designing a constant-volume combustion engine are to optimize the cycle by finding the best geometry, ensuring adequate air-fuel mixing, optimizing the expansion process of the nozzle through proper nozzle design, finding the best operating frequency, and controlling the valves to obtain this frequency. The valve timing must maximize thrust, yet allow sufficient residual gas to provide an ignition source for the next cycle. The focus of this study is to apply the results in a device for the propulsion source of miniaturized unmanned aerial vehicles, thus a fuel source must be chosen that is economical, compact, readily available, and carried in light-weight, low pressure fuel tanks.

1.2 Research Objectives

The objectives of this research are to find a simple, practical, low cost and efficient design of a possible constant volume combustion jet engine which can be used as a propulsion source of an aerospace vehicle. The device should allow high speed or low speed flight and have long range and loiter times when necessary, thus requiring a high turn-down ratio and high thrust or high specific impulse, as necessary. The following objectives will be pursued in this research:

1. Determine an engine geometry which provides high Specific Impulse ($I_{sp}$). $I_{sp}$ values above 3000 seconds are desired.

2. Determine an optimum operating frequency for this geometry which maximizes Specific Impulse and will provide the highest thrust for a given fuel flow.
3. Determine the inlet and throat valve timing which optimizes the Specific Impulse and provides the highest thrust for a given fuel flow.

4. Although this engine will be an unsteady device with a wide range of reservoir pressures, determine a nozzle design for the expansion phase which will provide the best possible thrust and specific impulse for the engine.

5. Find a means to properly mix the fuel and air in the combustion chamber, which will provide the desired performance with minimum fuel loss.

6. Determine the performance of the engine when it is operating in a convective airstream, over the entire range of the vehicle’s operating envelope, as it would on a vehicle in flight.

7. Analyze the thermodynamic cycle of the engine in order to properly predict its behavior and efficiency while in operation.

8. Determine the estimated engine and vehicle thrust-to-weight ratio.

It is important to note that specific impulse, which is a measure of the amount of thrust produced for the fuel used, is the driving factor in the design, since once an operating point which maximizes specific impulse is found, the geometry can then be scaled upward or downward to provide a higher or lower thrust, if desired, without changing the specific impulse. This ability to scale the engine as needed is a unique characteristic of this engine due to its extremely simple design. The target value of 3000 seconds for $I_{sp}$ is chosen to hopefully make it competitive with the more complicated and heavier turbojet engine. The chart below (Fig. 1-2) summarizes the range of specific impulses achieved by various
engines using hydrocarbon and hydrogen fuels [3]. The target for this propulsion engine, if achieved, will place it in the company of efficient turbojet engines and pulse-detonation engines (PDE’s).

![Figure 1-2: Specific Impulse vs. Mach Number for Various Propulsion Systems [3]](image)
1.3 Combustion and Fluid Mechanics Models

The combustion and fluid mechanics models used in this research are from a commercially available computational fluid dynamics (CFD) software package called ANSYS CFX. This is an extremely robust and sophisticated software package which is almost unlimited in its possible applications, but requires a thorough understanding of the models to apply them properly and understand the results and their limitations.

1.3.1 Combustion Model

Propane was the fuel of choice in this research, as it can be liquefied under pressure for carriage in flight, and it is stable, economical, and readily available. The use of propane also simplified the combustion model by not requiring analysis of multi-phase flows.

A 5-step propane-air reaction model, provided by the CFX package, was chosen to simulate the combustion of propane. The five reactions included are propane oxidation, carbon monoxide oxidation, hydrogen oxidation, and forward and backward water-gas reactions.

Also, the Eddy Dissipation Model was used to simulate the combustion process, since for this device the chemical reaction rate is fast relative to the transport processes in the flow.

In the eddy dissipation model, at the molecular level, when reactants mix, they instantaneously form products. It is assumed that the reaction rate may be related directly to the time required to mix the reactants at the molecular level. In turbulent flows, this mixing time is dominated by the eddy properties. Thus the reaction rate is proportional to a mixing time defined by the turbulent kinetic energy, $k$, and dissipation, $\varepsilon$. 
The concept of reaction control is applicable in many industrial combustion problems where reaction rates are fast compared to reactant mixing rates. Because of its simplicity and robust performance in predicting turbulent reacting flows, the eddy dissipation model has been widely applied in the prediction of industrial flames. [4][21]

1.3.2 Turbulence Model

To model the flow in the engine simulation, the $k$-$\varepsilon$ turbulence model was used. It has proven to be stable and numerically robust and has a well-established regime of predictive capability. For general-purpose simulations, the $k$-$\varepsilon$ model offers a good compromise in terms of accuracy and robustness. [5][20]

$k$ is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of $(L^2/T^2)$, e.g. m$^2$/s$^2$. $E$ is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of $k$ per unit time $(L^2 T^{-3})$, e.g. m$^2$/s$^3$.

The $k$-$\varepsilon$ model introduces two new variables into the system of equations. [5] The continuity equation is then:

$$\frac{d\rho}{dt} + \nabla \cdot (\rho \mathbf{U}) = 0$$

and the momentum equation becomes
\[ \frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) - \nabla \cdot (\mu_{\text{eff}} \nabla U) = \nabla p' + \nabla \cdot (\mu_{\text{eff}} \nabla U)^T + B \]

where \( B \) is the sum of body forces, \( \mu_{\text{eff}} \) is the effective viscosity accounting for turbulence, and \( p' \) is the modified pressure given by

\[ \mu_{\text{eff}} = \mu + \mu_t \]

\[ p' = p + \frac{2}{3} \rho k \]

where \( \mu_t \) is the turbulence viscosity. The \( k-\varepsilon \) model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation rate via the relation

\[ \mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \]

where \( C_\mu \) is a constant.

The values of \( k \) and \( \varepsilon \) come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate:
\[ \frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left( \frac{\mu}{\sigma_k} \nabla k \right) + P_k - \rho \varepsilon \]

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left( \mu \frac{1}{\sigma_\varepsilon} \nabla \varepsilon \right) + \frac{\varepsilon}{k} (C_{s1} P_k - C_{s2} \rho \varepsilon) \]

where, \( C_{s1}, \ C_{s2} \) and \( \sigma_k, \sigma_\varepsilon \) are constants. \( P_k \) is the turbulence production due to viscous and buoyancy forces, which is modeled using:

\[ P_k = \mu_l \nabla U \cdot (\nabla U + \nabla U^T) - \frac{2}{3} \nabla \cdot U (3 \mu_l \nabla \cdot U + \rho k) + P_{kb} \]

1.4 CFD Model Geometry

The initial geometry selected for this study is based upon that of a hobby-scale pulsejet. This university has completed a number of research studies on pulsejet operation, optimization, and miniaturization, analytically, numerically, and experimentally. The pulsejet is an unsteady propulsion device that generates intermittent thrust, with the first operational engine developed by the Germans in the 1930’s and known as the V1 or “buzz bomb”. Below in Fig. 1-3 is a schematic of that first pulsejet. [6]

The pulsejet used a series of reed or flapper valves at the intake end of the tube to take in a volume of fresh air to mix with the atomized fuel prior to ignition. Combustion of the fuel-air mixture initiates from residual hot gases still in the combustion chamber. As the combustion chamber pressure rises, the reed valve now acts as a check-valve, preventing the gases from escaping. Thrust is then generated from the expulsion of products down the long
rearward facing open-ended tube. After this expulsion, an expansion wave travels back forward toward the reed valve, causing it to open and bring in the fresh air, as the cycle starts again. The pulsejet is an uncontrolled, acoustic device that operates at one frequency, which is a function of the tail-pipe length (in the case of the V-1, this frequency was approximately 80 hz). It generates a single thrust and is generally an inefficient device. [6] [7] [8]

![Diagram of pulsejet engine](image)

*Figure 1-3: Schematic of German Aeropulse used to power V1 "buzz bomb"

The pulsejet geometry, however, is a good starting point for the geometry of the constant volume combustion jet engine of this research. Using the hobby scale pulsejet shown in Fig.
1-4 below, the long tailpipe is removed and replaced with a converging-diverging (de Laval) nozzle for expansion of the product gases.

Figure 1-4: Hobby Scale "Dyna-Jet" Pulsejet

The air inlet geometry used by the hobby-scale pulsejet will initially be maintained in the CFD model. The pulsejet inlet, as shown below (Fig. 1-5), is a series of ten small ports lying along the outer perimeter of the combustion chamber front wall, which open and close
passively with the acoustic cycle, via reed valves. The initial CFD model for this study will be a single annulus shaped inlet along the outer perimeter of the chamber wall. Later in this study, it will be necessary to determine a more practical inlet technique, an inlet that is compatible with an actively controlled valve.

Additionally, the constant volume combustion engine will utilize a commercial-off-the-shelf compressor, prior to the combustion chamber inlet, to improve overall pressure ratios and thus allow significantly higher thrust and specific impulse. However, using this mechanical compression will also result in higher costs and complexity. A compressor ratio of 2.5 is
chosen – a typical specification for a small, low cost compressor. This compression will be applied throughout operation - at static conditions and at all times when the supporting vehicle is in flight, experiencing a pressure rise from a convective air stream.

Below is depicted the initial model geometry (Fig. 1-6) with the perimeter air inlet to be used for CFD simulations for the constant volume engine.

Figure 1-6: Initial Geometry for Constant Volume Combustion Jet Engine CFD Simulations

Below in Figures 1-7 and 1-8 are also isometric and plan views of the proposed constant volume combustion engine.
Figure 1-7: Constant Volume Combustion Jet Engine Isometric View

Figure 1-8: Constant Volume Combustion Jet Engine Plan View
1.5 Model Operation

For the initial model, which will include static simulations only (no convective airstream), air from the compressor (1) to the inlet (2) will be maintained at 2.5 bar through a boundary condition at the compressor leading face of 2.5 bar. Inlet pressure will only momentarily fall below 2.5 bar during inlet valve (2) opening. For these early simulations, inlet valve opening will introduce a premixed stoichiometric propane-air mixture.

Determination of the duration of the inlet valve’s opening will be discussed later, but since the engine is still finishing the expansion phase with an open throat valve, this stage serves two purposes – introduce the fresh propane-air mixture to the combustion chamber (3) and to promote a purge of the last of the chamber products from the previous stage.

The throat valve (4) closes just prior to the closing of the inlet valve. With both the inlet and throat valves now closed, ignition of the reactants in the combustion chamber will now occur from the small amount of residual gases remaining in the combustion chamber. A summary of the valve operation is:

- Inlet valve closes at a pre-determined time and as chamber pressure approaches 2.5 bar.
- Reactants mix with residual hot products and combustion occurs.
- Due to isochoric combustion process, peak chamber pressures approach 10 bar, at peak temperatures of about 2500K.
- Throat valve opens and gases exhaust through the diverging nozzle.
- Due to high chamber pressure, supersonic exhaust velocities are possible.
• At a pre-determined time, as chamber pressure drops towards 1 bar, the inlet valve opens to provide some flushing of combustion chamber products and introduce new reactants to the combustion chamber.

• Soon after, the throat valve closes to allow capture of fuel-air mixture in the combustion chamber. Inlet valve closes again at approximately 2.5 bar chamber pressure and the cycle repeats.

The pulsejet’s exhaust tube was replaced with a nozzle designed for a reservoir pressure of 6 bar (6 x 10⁵ N/m²). This design was calculated using a MATLAB publically available software program (Appendix D). With the knowledge that this engine will be an unsteady device with large variations in combustion chamber pressure, 6.0 bar was chosen as the initial nozzle design point because peak chamber pressure was expected to be approximately 8 – 10 bar. With compression and expansion pressures approximating a sine curve, average combustion chamber pressure would be about 6 - 7 bar and using a 6.0 bar design point would minimize over-expansion and hence shocks in the nozzle. A nozzle optimization analysis will occur later in this study.

For initial simulations, the inlet and throat valves will be binary valves, open or closed, actively controlled. More realistic, controllable valves will be discussed and modeled later in this study.

1.6 CFX Model

ANSYS CFX is a general purpose Computational Fluid Dynamics (CFD) software suite that combines an advanced solver with powerful pre- and post-processing capabilities. [9][10] It includes the following features:
• An advanced coupled solver that is both reliable and robust.
• Full integration of problem definition, analysis, and results presentation.
• An intuitive and interactive setup process, using menus and advanced graphics.

ANSYS CFX is capable of modeling:
• Steady-state and transient flows
• Laminar and turbulent flows
• Subsonic, transonic and supersonic flows
• Heat transfer and thermal radiation
• Buoyancy
• Non-Newtonian flows
• Transport of non-reacting scalar components
• Multiphase flows
• Combustion
• Flows in multiple frames of reference
• Particle tracking.

ANSYS CFX consists of four software modules that take a geometry and mesh and pass the information required to perform a CFD analysis:
• CFX-Pre, is used to define simulations.
• CFX-Solver solves all the solution variables for the simulation for the problem specification generated in CFX-Pre.
• The CFX-Solver Manager module provides greater control to the management of the CFD task. Its major functions are:
• Specify the input files to the CFX-Solver.
• Start/stop the CFX-Solver.
• Monitor the progress of the solution.
• Set up the CFX-Solver for a parallel calculation.

• CFD-Post provides state-of-the-art interactive post-processing graphics tools to analyze and present the ANSYS CFX simulation results.

The model geometry as shown below is first built in the ANSYS Modeler. All simulations in this study were built in two-dimensions with an eight-degree symmetric wedge. Doing this was allowed because of the 2-D axisymmetric characteristics of the engine geometry. This greatly reduced the processing time for the simulations, which even with the wedge model, often approached 3 – 5 days in duration per engine cycle. Extrinsic properties such as fuel flow and thrust were computed from wedge values to engine values using software user expressions. The two images below (Fig. 1-9 and Fig. 1-10) depict the constant volume combustion engine and the surrounding outer domain.
Figure 1-9: Constant Volume Combustion Jet Engine CFD Wedge Model
The geometry is then meshed, again using the ANSYS Modeler, adjusting areas in the mesh that need refinement (Fig 1-11). Refinement controls include line and point controls, node spacing, inflation at boundaries (Fig 1-12), etc.
Figure 1-11: Constant Volume Combustion Jet Engine Wedge Model After Meshing

Figure 1-12: Mesh Inflation Control Used to Decrease Node Spacing in Boundary Layer
After placing the geometry in CFX-Pre, the initial conditions, boundary conditions, fuel and air parameters, fuel sources, and output parameters are prescribed. Also, expressions are built to control valves, fuel flow rates, etc.

The initial geometry boundary conditions are:

- **Outer Domain:** Air, 1 bar, 300K
- **Inlet Domain:** Stoichiometric Propane-Air Mixture, 2.5 Bar, 370K
- **Outer-Inlet Boundary:** 2.5 Bar, 370K, Propane-Air Mixture
- **All Walls:** Adiabatic, No-Slip, Smooth Wall (Outer, Inlet, Combustion Chamber, Nozzle)

The combustion model designated throughout these simulations is summarized as:

- Propane Air Mixture – Inlet (Premixed) at 2.5 Bar, 370K
- Combustion Model – Five-Step, Eddy Dissipation
- Chemical Timescale – $2.5 \times 10^{-4}$ seconds (Neutral setting of three available)
- Extinction Temperature – 900K in the combustion chamber
- Extinction Temperature – 3000K in all domains except the combustion chamber
- Heat Transfer Model – Total Energy

The fluid mechanics model used is summarized as:

- Turbulence Model – $\kappa$-Epsilon
- Turbulent Wall Functions – Scalable
- Heat Transfer Model – Total Energy
2.1 Valve Control Background

The only active control of the initial constant volume combustion jet engine occurs with the inlet and throat valves. Later in this study, fuel flow rates will be added as a means of active control. The initial CFX simulations need to determine when the inlet and throat valve should open and close and the length of time that the valves should be open and closed. These initial CFX simulations will use a trial and error method, observing when reliable combustion will occur for a cycle and what values will yield maximum pressures, and thus maximum thrust. The knowledge determined about the valve timing and sequencing will be specific to the geometry used in the simulations, however, it will also apply to all geometries that are scaled up or down from the simulation geometry.

Optimizing the inlet and throat valve sequencing can be achieved by varying the following and noting the effects on combustion chamber peak pressure and thrust produced:

1) Expansion - Throat Open, Inlet Closed
2) Flushing – Throat Open, Inlet Open
3) Recharge – Throat Closed, Inlet Open
4) Combustion – Throat Closed, Inlet Closed
All simulations will begin with an initial condition in the combustion chamber of a pressure of $6.0 \times 10^5$ N/m$^2$ and a temperature of 2500K, with both the inlet valve and the throat valve in the closed position. Shortly after the simulation begins, the throat valve will open to begin the first expansion phase. Starting the first cycle with this initial condition serves as a “jump-start” to provide the residual gases needed for the next cycle. In the actual engine, when built, this initial process would be substituted with an ignition source, such as a spark plug, for the first cycle.

As the expansion continues and the combustion chamber pressure approaches the nozzle back-pressure, in this case atmospheric pressure of 1.0 bar, the inlet valve will open to introduce the fresh fuel-air mixture at 2.5 bar. This valve opening will aid in flushing residual gases from the combustion chamber and provide new reactants for the next cycle. The throat valve will close slightly prior to the inlet valve closure in the initial simulations to provide a definitive recharge phase where reactants and residual gases are captured together. Finally, the inlet valve will close to begin the combustion or compression phase. It is desired to continue the compression until maximum combustion chamber pressure is attained, without any stagnation at this maximum pressure. Holding the peak pressure serves no benefit and extends the cycle time, which reduces average thrust. When peak pressure is reached, the throat valve opens to begin the next cycle.

Control of the opening and closing of the inlet and throat valve in ANSYS CFX is accomplished through a Resistance Loss Coefficient (m$^{-1}$). [11] A Loss Coefficient of zero allows full flow through the valve, while a high Loss Coefficient, usually on the order of $10^6$ m$^{-1}$ through $10^8$ m$^{-1}$, will prevent all flow through the valve, even under extremely high
pressures. Application of the Resistance Loss Coefficient, including timing and duration of
the opening and closure, is accomplished through CFX expressions written by the user – the
depiction below (Fig. 2-1) shows the Loss Coefficients as used in this model for the two
binary inlet (red) and throat (green) valves. The timesteps used are $10^{-6}$ seconds.

![Diagram showing the Loss Coefficient for Control of Binary Inlet and Throat Valves]

*Figure 2-1: Resistance Loss Coefficient for Control of Binary Inlet and Throat Valves*

A plot of typical combustion chamber pressure is shown below (Fig. 2-2), along with other
pressure monitor points and their locations. These monitor points will be used throughout
this study.
2.2 Valve Timing Observations

While attempting to maximize pressure and thrust, it was observed that the length of the expansion phase is critical in maximizing the peak pressure in the next cycle. It is imperative that the combustion chamber pressure be reduced as close as possible to atmospheric pressure, or as a minimum, at least one bar below the inlet pressure (1.5 bar in this case). Expanding the gases as close as possible to atmospheric pressure apparently allows much better flushing of the products from the combustion chamber when the inlet opens, however it is also important that enough products then remain as an aid in combustion of the fresh reactants. Not attaining this target minimum pressure during expansion leaves too much residual gas for proper filling of the combustion chamber with fresh reactants. The chart

![Figure 2-2: Typical Pressure Plots with Monitor Point Locations Shown](image-url)
below (Fig. 2-3) shows the pressure differential between the inlet pressure and the minimum expanded chamber pressure versus the peak combustion chamber pressure attained in the subsequent cycle.

![Graph showing pressure differential](image)

**Figure 2-3: Effect of Minimum Expansion Pressure on Next Cycle’s Maximum Pressure**

Below (Fig. 2-4) are the results of trial simulations which attained a maximum combustion chamber pressure of approximately 13 bar. Total time for one cycle was 5150 timesteps or $5.15 \times 10^3$ seconds. This equates to a frequency of 194Hz. The ratios ($\tau$) of each phase to the total cycle time are also shown.
Total Timesteps Per Cycle: 5150 Timesteps = $5.15 \times 10^{-3}$ sec  Frequency = 194hz

Inlet Valve Open: 1000 Timesteps Total
Inlet Valve Closed: 4150 Timesteps Total

Throat Valve Open: 3150 Timesteps Total
Throat Valve Closed: 2000 Timesteps Total

**Sequence**

1) Expansion-Throat Open, Inlet Closed  \[ \tau_{\text{Exp}} = \frac{2600}{5150} = 0.51 \]

2) Flushing – Throat Open. Inlet Open  \[ \tau_{\text{Flush}} = \frac{570}{5150} = 0.11 \]

3) Recharge–Throat Closed, Inlet Open  \[ \tau_{\text{Charge}} = \frac{430}{5150} = 0.08 \]

4) Combustion–Throat Closed, Inlet Closed  \[ \tau_{\text{Comb}} = \frac{1550}{5150} = 0.30 \]

**Figure 2-4:** Time and Time Ratios Required for Each Cycle Phase, from Trial Simulations

Below is a pressure and thrust plot of the initial condition cycle and the next full cycle:

**Figure 2-5:** Pressure and Thrust Plots of Initial Condition Cycle and Next Full Cycle (Wedge)
Numerous simulations were also accomplished to determine the optimum duration of time to allow the inlet valve to remain open. As the chart below shows, as the inlet remained open beyond 1000 timesteps, or $1 \times 10^{-3}$ seconds, the specific impulse dropped off precipitously. This was due to fuel-air mixture being lost through the open throat valve at a much higher rate. When the inlet closed sooner than 1000 timesteps, the lower amount of fuel-air mixture entering the chamber resulted in lower maximum pressure and thrust values, also lowering specific impulse. Thus an inlet open duration of $1 \times 10^{-3}$ seconds was chosen as the optimum.

![Figure 2-6: Effect of Inlet Open Duration on Specific Impulse for 194hz Engine](image)

Figure 2-6: Effect of Inlet Open Duration on Specific Impulse for 194hz Engine
2.3 Conclusions

a. For this engine geometry, the optimum cycle time is $5.15 \times 10^{-3}$ seconds or 194 hz.

b. The optimum inlet open time (flushing and recharge) is 1000 timesteps, or $1 \times 10^{-3}$ seconds.

c. A cycle expansion time of 2600 timesteps, or $2 \times 10^{-3}$ seconds and a compression time of 1550 timesteps, or $1.55 \times 10^{-3}$ seconds is desired for maximum thrust.
Chapter 3  Perimeter Inlet Geometries and Simulations

3.1  Introduction

Geometry variations will now be simulated in order to find possible thrust and specific impulse improvements.

3.2  Single Inlet Full Combustion Chamber

The initial simulations will use the geometry below, as described in Section 1.4, based upon the hobby-scale pulsejet geometry. This geometry uses a single annulus shaped inlet along the outer perimeter of the combustion chamber front wall. Total inlet area is 1.822 in\(^2\). The combustion chamber size and throat are identical to the hobby-scale pulsejet, with a combustion chamber volume of 17.5 in sq and the nozzle area ratio is 1.75 with a length of 2.39 inches. The nozzle design point is for a reservoir of 6.0 x 10\(^5\) N/m\(^2\) and 2500K with gas density being stoichiometric propane-air.

![Single Inlet Full Combustion Chamber Geometry](image)

Figure 3-1:  Single Inlet Full Combustion Chamber Geometry
Below (Fig. 3-2) are simulation results for the single inlet full combustion chamber engine model. Pressures, thrust and fuel flow through the inlet are shown.

**Figure 3-2:** Single Inlet Full Combustion Chamber Pressure, Thrust and Fuel Flow Plots (Wedge)
Below are sequential temperature images of one cycle of the 194hz simulation for this model.

Figure 3-3 Sequential Temperature Images of One Cycle of the Single Inlet Full Combustion Chamber Engine Model
3.3 Dual Inlet Full Combustion Chamber

The initial geometry is now modified from a single inlet to a dual inlet in order to investigate possible improvements in combustion of the entering fuel-air mixture and to explore potential reductions of fuel losses through the nozzle. The additional inlet area is an annulus of area 0.555 in² for a total inlet area now of 2.375 in². This dual inlet will possibly alter the fuel-air mixture flow in a positive way to reduce fuel losses and improve the reactant mixing with residual gases. This revised geometry is shown below (Fig. 3-4).

![Figure 3-4: Dual Inlet Full Combustion Chamber Geometry](image)

Below (Fig. 3-5) are simulation results for the dual inlet full combustion chamber engine model. Pressure, thrust and fuel flow through the inlet are shown.
Figure 3-5: Dual Inlet Full Combustion Chamber Pressure, Thrust and Fuel Flow Plots (Wedge)
Below are sequential temperature images of one cycle of the 194hz simulation for this model.

Figure 3-6: Sequential Temperature Images of One Cycle of the Dual Inlet Full Combustion Chamber Engine Model
3.4 Single Inlet Reduced Combustion Chamber

The single inlet full combustion chamber geometry is now modified to a smaller combustion chamber to further explore possible improvements in combustion of the entering fuel-air mixture and to possibly reduce fuel losses through the nozzle. This revised geometry is shown below. The original full combustion chamber longitudinal dimension of 1.9 inches, prior to the convergence, is now reduced by 50% to 0.95 inches. This reduces the total chamber volume by 15% to 14.85 in sq. Nozzle geometry remains unchanged.

![Figure 3-7: Single Inlet Reduced Combustion Chamber Geometry](image)

Below (Fig. 3-8) are simulation results for the single inlet reduced combustion chamber engine model. Pressure, thrust and fuel flow through the inlet are shown.
Figure 3-8: Single Inlet Reduced Combustion Chamber Pressure, Thrust and Fuel Flow Plots (Wedge)
Below are sequential temperature images of one cycle of the 194hz simulation for this model.

Figure 3-9: Sequential Temperature Images for One Cycle of the Single Inlet Reduced Combustion Chamber Engine Model
3.5 Dual Inlet Reduced Combustion Chamber

The dual inlet full combustion chamber geometry is now also modified to a smaller combustion chamber to explore further possible improved combustion and reduced fuel losses. The original full combustion chamber longitudinal dimension of 1.9 inches, prior to the convergence, is now reduced to 0.95 inches. This reduces the chamber volume by 15% to 14.85 in sq. Nozzle geometry remains unchanged.

![Figure 3-10: Dual Inlet Reduced Combustion Chamber Geometry](image)

Below (Fig. 3-11) are simulation results for the dual inlet reduced combustion chamber engine model. Pressure, thrust and fuel flow through the inlet are shown.
Figure 3-11: Dual Inlet Reduced Combustion Chamber Pressure, Thrust and Fuel Flow Plots (Wedge)
Below are sequential temperature images of one cycle of the 194hz simulation for this model.

Figure 3-12: Sequential Temperature Images of One Cycle Of the Dual Inlet Reduced Combustion Chamber Engine Model
3.6 Single Inlet Reduced Combustion Chamber and Reduced Throat

It is now of interest to determine if improved thrust and reduced fuel losses may be obtained by reducing the converging-diverging nozzle throat size. The original throat size, as stated previously, was based upon the hobby-scale pulsejet tailpipe size. Of course, being an acoustic device with only a convergence and operating at much lower pressures, this tailpipe diameter is most likely too large for a converging-diverging nozzle. Therefore, the throat area used previously will now be reduced by 50%, from 1.1323 in$^2$ to 0.5566 in$^2$. This area reduction will also require changes to the nozzle exit area and length in order to maintain the nozzle design point for a 6.0 bar reservoir. These changes are shown in the depiction below.

![Diagram of Single Inlet Reduced Combustion Chamber and Reduced Throat Geometry]

Figure 3-13: Single Inlet Reduced Combustion Chamber and Reduced Throat Geometry
Below (Fig. 3-14) are simulation results for the single inlet reduced combustion chamber and reduced throat engine model. Pressure, thrust and fuel flow through the inlet are shown.

Figure 3-14: Single Inlet Reduced Combustion Chamber and Reduced Throat Pressure, Thrust and Fuel Flow Plots (Wedge)
Below are sequential temperature images of one cycle of the 194hz simulation for this model.

![Sequential Temperature Images](image)

**Figure 3-15:** Sequential Temperature Images of One Cycle of the Reduced Combustion Chamber and Reduced Throat Engine Model

Analysis of the average thrust, average fuel flow and specific impulse of this 194hz model will be accomplished in the next chapter as comparisons are made of this frequency and these different geometries with other operating frequencies.
4.1 Method of Frequency Control

The analysis to this point has used a cycle time of \(5.15 \times 10^{-3}\) seconds or 194 hz, which was determined to be ideal for the geometry being modeled. Now it is necessary to explore the possibility that other higher frequencies may provide the benefit of even higher specific impulse. There is no need of course to explore lower frequencies, due to the previous discovery that 194hz provides a peak combustion chamber pressure, and thus a peak thrust, without stagnating at that pressure – any frequency lower than 194hz would require stagnation at that peak pressure, lowering average thrust and specific impulse. The previously determined 194hz cycle phases and lengths are shown again below.

194hz Cycle Phases

1) Expansion-Throat Open, Inlet Closed  \(\tau_{\text{exp}} = \frac{2600}{5150} = 0.51\)

2) Flushing – Throat Open, Inlet Open  \(\tau_{\text{flush}} = \frac{570}{5150} = 0.11\)

3) Recharge–Throat Closed, Inlet Open  \(\tau_{\text{charge}} = \frac{430}{5150} = 0.08\)

4) Combustion–Throat Closed, Inlet Closed  \(\tau_{\text{comb}} = \frac{1550}{5150} = 0.30\)

The increased frequencies will be accomplished through adjustments to the actively controlled inlet and throat valves. The total cycle time of 5150 timesteps will be reduced to provide new frequencies of 223hz, 259hz, 310hz and 388hz. The previously determined phase lengths for flushing (570 timesteps) and recharge (430 timesteps) will be maintained,
as they are critical for the proper charging and ignition of the cycle. Of the remaining
timesteps for each frequency, the ratio of expansion to combustion (0.51/0.30 or 1.7:1) will
be maintained as the total combustion and expansion time is decreased to meet the desired
increased frequency. Many simulations were accomplished and the results are described
below for each of the five previously discussed geometries.

4.2 Single Inlet Full Combustion Chamber
Plots of the average thrust, average fuel flow and specific impulse for the single inlet full
combustion chamber model for each frequency are shown below.

\[\text{Figure 4-1: Thrust vs. Frequency for the Single Inlet Full Combustion Chamber Model}\]
Figure 4-2: Fuel Flow vs. Frequency for the Single Inlet Full Combustion Chamber Model
Figure 4-3: Specific Impulse vs. Frequency for the Single Inlet Full Combustion Chamber Model

4.3 Dual Inlet Full Combustion Chamber

Plots of the average thrust, average fuel flow and specific impulse for the dual inlet full combustion chamber model for each frequency are shown below.
Figure 4-4: Thrust vs. Frequency for the Dual Inlet Full Combustion Chamber Model

Figure 4-5: Fuel Flow vs. Frequency for the Dual Inlet Full Combustion Chamber Model
4.4 Single Inlet Reduced Combustion Chamber

Plots of the average thrust, average fuel flow and specific impulse for the single inlet reduced combustion chamber model for each frequency are shown below.

Figure 4-6: Specific Impulse vs. Frequency for the Dual Inlet Full Combustion Chamber Model
Figure 4-7: Thrust vs. Frequency for the Single Inlet Reduced Combustion Chamber Model

Figure 4-8: Fuel Flow vs. Frequency for the Single Inlet Reduced Combustion Chamber Model
4.5 Dual Inlet Reduced Combustion Chamber

Plots of the average thrust, average fuel flow and specific impulse for the dual inlet reduced combustion chamber model for each frequency are shown below.

Figure 4-9: Specific Impulse vs. Frequency for the Single Inlet Reduced Combustion Chamber Model
Figure 4-10: Thrust vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model

Figure 4-11: Fuel Flow vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model
Figure 4-12: Specific Impulse vs. Frequency for the Dual Inlet Reduced Combustion Chamber Model

4.6 Single Inlet Reduced Combustion Chamber and Reduced Throat

Plots of the average thrust, average fuel flow and specific impulse for the single inlet reduced combustion chamber and reduced throat model for each frequency are shown below.
Figure 4-13: Thrust vs. Frequency for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model

Figure 4-14: Fuel Flow vs. Frequency for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model
Figure 4-15: Specific Impulse for the Single Inlet Reduced Combustion Chamber and Reduced Throat Model

Below for comparison purposes is Figure 4-16, a composite plot of all studied geometries and their resultant specific impulse values as a function of varied operating frequencies.
4.7 Conclusions

Inspecting the results of the frequency analysis, it is clear that the best performance occurs with the single inlet, reduced combustion chamber and reduced throat model, operating at 194 hz. At this operating point, average thrust is 51 Newtons, average fuel flow is 1.92 grams per second, and specific impulse is 2700 seconds. All simulations from this point forward in this study will operate at 194 hz with a single air inlet. The reduced combustion chamber and reduced throat engine geometry will remain unchanged also, except for a revision to the inlet air location in Chapter 6 and possible nozzle geometry improvements in Chapter 9.
Chapter 5  Reactant Mixing

5.1  Introduction

Up to this point, all simulations have utilized a boundary condition at the compressor, which provides a stoichiometric, premixed air-fuel mixture to the combustion chamber inlet. This is, of course an optimum, but unrealistic condition. Therefore, it is now necessary to look at ways to successfully mix the propane with the incoming air, while in the combustion chamber. This will be accomplished through injection of the fuel at one or two locations, usually in or near the inlet air stream. In addition, the fuel will be injected into, away from or perpendicular to the incoming airstream.

It is also desired to minimize fuel losses through the nozzle while the combustion chamber is being recharged with reactants. These fuel losses can be reduced or eliminated in two ways – (1) promote reactant mixing in an area of the combustion chamber distant from the throat and (2) possibly shorten the time that the throat is open when the inlet is open. Adjustments to the inlet and throat valve timing are also desired to eliminate any backflow from the combustion chamber, through the inlet, while the inlet is still open. Previous simulations have shown this to sometimes occur when the throat closes too soon before the inlet closes. Single and dual fuel injection points will be explored, at total flow rates of 1.5 g/s, 2.0 g/s, 2.5 g/s and 3.0 g/s. The effects on thrust and specific impulse will be evaluated.
5.2 Single Fuel Injection

This simulation will no longer include a premixed stoichiometric air-fuel mixture entering the combustion chamber. At static conditions, standard outside air compressed to \(2.5 \times 10^5\) N/m\(^2\) from the compressor enters the combustion chamber. A single propane source point is located in the combustion chamber air stream as it enters. The propane flows steadily from the source point only when the inlet valve is open. Both the inlet air valve and the fuel inlet valve are binary states (full open or full closed), as in previous simulations.

For the single fuel inlet simulations, fuel enters the combustion chamber directly into the inlet air stream and four different cases were modeled – fuel with no initial velocity, fuel at 50 m/s into the air stream, fuel at 50 m/s away from the air stream, and fuel at 50 m/s perpendicular from the airstream in a downward direction. Sequential images of one cycle of the propane mass fraction for the single point fuel injection with no initial velocity are shown below. As can be seen from the visual images, fuel losses through the open throat valve are minimal.
5.3 Dual Fuel Injection

Three cases of dual fuel injection were modeled – in all cases one fuel injection was directly in the incoming air stream and the other fuel injection was near the combustion chamber centerline axis, below the first injection point, in the clockwise air flow vortex. The amounts of fuel at each injection point (centerline fuel/airstream fuel) in these three simulations were:

- 1.5 grams per sec/0.5 grams per sec  (2.0 grams per sec total)
- 1.125 grams per sec/0.375 grams per sec  (1.5 grams per sec total)

Figure 5-1: Sequential Images of Propane Mass Fraction During One Cycle of Single Fuel Inlet, No Initial Fuel Velocity
- 1.0 grams per sec/1.0 grams per sec  (2.0 grams per sec total)

Sequential images of one cycle of the propane mass fraction for the dual point fuel injection are shown below for the case of 1.5 grams/sec centerline fuel injection and 0.5 grams/sec airstream fuel injection. From the visual images (Fig. 5-2), it is clear that fuel losses still occur through the throat valve and some losses also now occur from centerline injected fuel flowing back through the air inlet.

Thrust and specific impulse are plotted below for all cases for comparison. There is little significant performance distinction between the single and dual fuel injection cases, although a slight improvement in thrust exists when the single injection has a perpendicular downward velocity and also when the dual fuel injection is divided 1.125 grams/sec near the centerline and 0.375 grams/sec in the airstream. There also appears to be reduced specific impulse for the dual fuel injection cases, most likely due to higher fuel losses through the open throat, and at times, through the still-open inlet.
Figure 5-2: Sequential Images of Propane Mass Fraction During One Cycle With 1.125 Grams/Sec Near Centerline and 0.375 Grams/Sec In Airstream

Below (Figures 5-3 and 5-4) are plots of the results of the reactant mixing simulations:
Figure 5-3: Thrust vs. Average Fuel Flow, Single and Dual Injection, Various Locations and Velocities
Figure 5-4: Specific Impulse vs. Fuel Flow, Single and Dual Injection, Various Locations and Velocities
5.4 Conclusions

5.4.1. There are no distinct advantages from using a dual fuel injection scheme or a single fuel injection scheme with initial fuel velocities, especially when considering the added engine complexities inherent in those cases. Therefore the single fuel injection in the airstream, with no initial velocity should be used. It is also evident that closing the throat valve and inlet at the same time should be modeled in future simulations to aid in reducing fuel losses.

5.4.2 There was no reduction in performance when comparing the single injection fuel mixing model with the premixed fuel-air model.

5.4.2.1 The premixed fuel-air model at 194 hz produced average thrust of 51 Newtons, at an average fuel flow of 1.92 grams per second, and had a specific impulse of 2700 seconds.

5.4.2.2 The single injection fuel mixing model produced 54 Newtons of thrust at a specific impulse of 2700 seconds. Also, a maximum thrust of 62 Newtons was produced at 3.0 grams/sec propane, at the minimum specific impulse of 2100 seconds. A thrust of 31 Newtons occurred at 1.0 grams/sec at the maximum specific impulse value of 3200 seconds.
Chapter 6  Valve Sinusoidal Area Flow Profiles and Single Centerline Air Inlet

6.1  Introduction

Previous simulations have modeled the inlet as a single annulus shaped opening around the circumference of the combustion chamber front wall. It is now necessary to relocate the inlet valve to a single ball or globe type valve, which is able to rotate at the desired cycle rate. Locating the valve to the longitudinal axis is desired in order to place it in-line with the throat valve, which will make possible simultaneous, synchronized drive and control of the inlet valve and throat valve. Placing the inlet valve on the longitudinal axis also simplifies, shortens and lightens the engine and thus reduces fabrication time and costs. An added benefit of this inlet valve location is that the simulations of this study are simplified, as the engine is still axisymmetric, which allows the continued use of an eight–degree wedge model in the CFX software.

In addition to relocating the inlet valve, and modeling a ball or globe valve, it will be necessary to model the mass flow through the inlet and the throat valves as sinusoidal flow profiles to match the sinusoidal area profiles of the actual globe valves as they rotate. This will be accomplished through user-written expressions and the resistance loss coefficient, which was also used for the binary valves, but will now be modified to model the sinusoidal flow profiles.

6.2  Inlet Valve Relocation

As seen in the depictions below, the relocated inlet valve is now a single, circular valve, centered on the face of the combustion chamber. The area of the full opening of this valve
will be sized to match the inlet area of the previous simulations, 1.82 in\(^2\). Although this area may appear too small at first glance, since the globe valve throttling will produce a smaller total area profile over time than the binary valve, it was chosen as a first estimate for the model in an attempt to reduce the overall size of the inlet valve. The size of the globe valve is chosen to provide the timing that was determined in Section 2.1, i.e., when the valve is rotating, it will be partially or fully open \(1 \times 10^{-3}\) seconds and fully closed \(4.15 \times 10^{-3}\) seconds. Also note that the 194hz cycle will only require that the inlet and throat valves rotate at one-half of that value, or 97hz, since the globe valves are symmetric and one cycle will occur through one-half of each valve’s rotation.

A comparison of the previous binary valve model with the new model with a centrally located inlet valve is shown below.

![Diagram](image.png)

*Figure 6-1: Constant Volume Combustion Jet Engine with Annulus Shaped Air Inlet On Combustion Chamber Front Wall, As Used in Initial Simulations*
6.3 Valve Model in CFX

When modeling the sinusoidal flow profile of a rotating globe valve using the resistance loss coefficient, simulation results showed that it was necessary to lower the “closed position” loss coefficient to approximately $1 \times 10^4 \text{ m}^{-1}$ in order to properly follow the sine functions. For comparison purposes, the binary valve loss coefficient used was $1 \times 10^8 \text{ m}^{-1}$. However, when using this lower resistance loss coefficient, the closed position was found to be porous under very high pressures. To correct this condition, in the model only, a binary valve is added in series with the sinusoidal globe valve. As the sinusoidal globe valve reaches the closed position, the binary valve also closes to provide a positive seal with no mass allowed through the valve. As the globe valve begins to open, the binary valve opens fully, and the sinusoidal flow profile is followed during the opening.

Figure 6-2: Constant Volume Combustion Jet Engine with Inlet Air Valve Centered on Engine Longitudinal Axis
This globe valve model will also be used to model the throat valve, again using the sinusoidal valve and binary valve in series, in the CFX model only. The two valves in series can be seen in Figure 6-2 at both the inlet and throat. The depictions below show comparisons of typical results for the sinusoidal valve mass flow and the previously used binary valve mass flow for the inlet and throat valves.

*Figure 6-3: Comparison of Air Mass Flow Through Inlet Using Binary and Sinusoidal Flow Models (Wedge)*

*Figure 6-4: Comparison of Mass Flow Through Throat Using Binary and Sinusoidal Flow Models (Wedge)*
Below is the depiction of thrust using a sinusoidal globe valve, as compared to the previously used binary valve.

![Figure 6-5: Comparison of Thrust Using Binary and Sinusoidal Throat Valve (Wedge)](image)

6.4 Sinusoidal Fuel Flow Profile

Since it was previously found that the best fuel mixing occurs when the fuel is released with no initial velocity into the incoming air stream, it is also necessary to match the incoming fuel mass flow rate with the incoming air mass flow rate. In the previous binary inlet valve models, fuel mass flow was also introduced into the combustion chamber as a constant binary function. With a sinusoidal air inlet flow profile, fuel mass will now also be introduced to the combustion chamber as a sinusoidal mass flow, with the fuel flow occurring only when the air inlet valve is open. Fuel flow control was accomplished by adjusting the peak sine function command to the wedge model in CFX and then the fuel flow
was calculated by averaging the fuel flow over the cycle and the entire engine. A typical depiction of the fuel mass flow is shown below.

![Sinusoidal Valve Area Flow Profiles](image)

*Figure 6-6: Sinusoidal Fuel Flow Profile Used to Match the Sinusoidal Air Inlet Profile (Wedge)*

In Figures 6-7 and 6-8 are a conceptual exploded plan profile and an isometric view of the general configuration of the inlet and throat valves, along with the combustion chamber and nozzle. As each globe valve rotates, the flow is throttled from full closed to full open, and then full closed again as the valve opening passes the stationary “face” at the combustion chamber and nozzle openings. The opening of the inlet valve is 1.82 in$^2$ and the opening of the throat valve is 0.55 in$^2$. The radius of each valve is calculated by the ratio of valve open time or partially open time to valve closed time. This results in an inlet valve radius of 2.42 in and the radius of the throat valve being 0.48 in.
Figure 6-7: *Inlet Valve and Throat Valve Exploded Plan View*

Figure 6-8: *Inlet Valve and Throat Valve Exploded Isometric View*
As mentioned previously, the symmetry of each globe valve allows each valve to rotate at 97hz, one-half of the 194hz cycle frequency, as one half-rotation includes one valve open phase and one valve closed phase. A battery-powered direct-drive synchronous electric motor can be used to drive both valve rotations at the 97hz speed through a mechanical linkage. An aerodynamic sleeve can be used to cover the inlet valve and drive linkage to the throat valve, similar to the depiction below.

![Diagram of constant volume combustion engine with sleeve covering inlet valve]

**Figure 6-9: Constant Volume Combustion Engine with Sleeve Covering Inlet Valve**

### 6.5 CFX Simulations with Inlet and Throat Valves Modeled with Sinusoidal Flow Profiles

It is necessary to determine the effect of the valves’ sinusoidal mass flow profiles on the engine performance, in particular the thrust and specific impulse. It is expected that engine performance will be adversely affected somewhat due to the throttling through the inlet and throat valves. Throttling through the valves increases the already unsteady flow characteristics of this engine.
Early simulations found that the new centralized air inlet was much more efficient in removing residual gases from the combustion chamber, to the degree that it became necessary to reduce the inlet open duration by 10% (1000 timesteps to 900 timesteps) in order to leave enough residual gas in the combustion chamber for initiation of the subsequent cycle. At the same time, with this improved inlet air efficiency, the staggered closing of the throat valve and then the inlet valve became unnecessary, and these two valves are now closed simultaneously.

The accomplished simulations are summarized in the table below. Modeled fuel flows range from 0.43 grams per second to 2.43 grams per second. Combustion at fuel flow rates below 0.43 grams/second was unsustainable. The peak thrust of 44.8 Newtons was attained at 2.28 grams/second of propane, which appears to be the engine’s stoichiometric value at these static conditions – above this fuel flow rate, the thrust remains constant or decreases slightly. Also indicative of stoichiometry is the temperature attained of approximately 2600K, which approximates the constant volume adiabatic flame temperature of a propane-air mixture.

Also shown are plots of thrust and specific impulse as a function of fuel flow rate. Notice that thrust reaches a maximum value of 44.8 Newtons at a fuel flow rate of 2.28 grams/second and when fuel flow rates are below 0.86 grams/second, the thrust drops much more quickly. Thus, the specific impulse reaches its maximum of 3216 seconds at 0.86 grams/second of propane.
Table 6-1: Simulation Results Using Relocated, Centralized Valve Sinusoidal Flow Profiles

### Constant Volume Combustion Jet Engine
#### Sinusoidal Valve Flow Profile Results

<table>
<thead>
<tr>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newtons)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.43</td>
<td>15</td>
<td>3.1</td>
<td>1370</td>
<td>5.2</td>
<td>1240.7</td>
</tr>
<tr>
<td>0.71</td>
<td>50</td>
<td>4.2</td>
<td>1580</td>
<td>17.4</td>
<td>2729.5</td>
</tr>
<tr>
<td>0.86</td>
<td>70</td>
<td>5.1</td>
<td>1750</td>
<td>24.4</td>
<td>3216.6</td>
</tr>
<tr>
<td>1.00</td>
<td>88</td>
<td>5.7</td>
<td>1855</td>
<td>30.7</td>
<td>3032.8</td>
</tr>
<tr>
<td>1.28</td>
<td>100</td>
<td>6.5</td>
<td>2000</td>
<td>33.0</td>
<td>2757.1</td>
</tr>
<tr>
<td>1.57</td>
<td>118</td>
<td>7.2</td>
<td>2260</td>
<td>36.6</td>
<td>2522.1</td>
</tr>
<tr>
<td>1.71</td>
<td>119</td>
<td>7.5</td>
<td>2455</td>
<td>38.1</td>
<td>2392.3</td>
</tr>
<tr>
<td>1.86</td>
<td>119</td>
<td>7.3</td>
<td>2580</td>
<td>40.6</td>
<td>2290.5</td>
</tr>
<tr>
<td>2.00</td>
<td>127</td>
<td>7.8</td>
<td>2245</td>
<td>43.1</td>
<td>2188.4</td>
</tr>
<tr>
<td>2.14</td>
<td>138</td>
<td>8.7</td>
<td>2570</td>
<td>44.1</td>
<td>2092.6</td>
</tr>
<tr>
<td>2.28</td>
<td>132</td>
<td>9.1</td>
<td>2500</td>
<td>44.8</td>
<td>1990.3</td>
</tr>
<tr>
<td>2.43</td>
<td>132</td>
<td>8.8</td>
<td>2655</td>
<td>42.2</td>
<td>1766.1</td>
</tr>
</tbody>
</table>
Figure 6-10: Thrust vs. Average Fuel Flow for Model with Centralized Inlet and Sinusoidal Flow Valves
Figure 6-11: Specific Impulse vs. Average Fuel Flow for Model with Centralized Inlet and Sinusoidal Flow Valves
Below is a sequential set of propane mass fraction images during one cycle of the high thrust case, with average fuel flow of 2.28 grams per second. Important to notice from these images is the much improved fuel-air mixing that occurs with the inlet valve relocation to the longitudinal axis. The engine geometry and air inlet location create a passive counterclockwise vortex just above the fuel entry point which aids in fuel-air mixing and is also the primary reason for reduced fuel losses through the throat prior to throat closure.

Figure 6-12: Sequential Images of Propane Mass Fraction During One Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves
Below is a sequential set of images of the engine temperature during one cycle of the maximum thrust case, which occurs at an average fuel flow rate of 2.28 grams per second.

Figure 6-13: Sequential Images of Temperature During One Maximum Thrust Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves
The following sequential temperature images are of one cycle of the engine while operating at a high specific impulse fuel flow rate of 1.28 grams per second. Of note are the reduced temperatures, but still effective mixing and combustion for adequate thrust production.

Figure 6-14: Sequential Images of Temperature During One Maximum Thrust Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves (continued)

Figure 6-15: Sequential Images of Temperature During One High Specific Impulse Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves
Figure 6-16: Sequential Images of Temperature During One High Specific Impulse Cycle of Model with Centralized Inlet and Sinusoidal Flow Valves (continued)
6.6 Conclusions

6.6.1. Peak thrust when the valves are modeled using a sinusoidal mass flow profile occurs at 2.28 grams per second at approximately 45 Newtons of thrust. Comparing this to the peak thrust obtained with binary valves, modeled in Chapter 5, which occurred at 3 grams per second and approximately 62 Newtons, there is a 28% reduction in peak thrust production when using the globe valves. However, the relocation of the air inlet to the longitudinal axis and simultaneous closing of the inlet and throat valves, together improved fuel-air mixing, reduced fuel losses through the throat valve, and allowed the peak thrust to be obtained at 24% less average fuel flow. This results in maintaining a peak specific impulse of over 3200 seconds with the sinusoidal mass flow profiles, or globe valves.

6.6.2. From the thrust and specific impulse plots, a rewarding find of the simulations is the outstanding turn-down capability of the engine - approximately 5:1. Peak thrust occurs at 2.28 grams per second and peak specific impulse occurs at 0.43 grams per second. This provides a propulsion system capable of providing long range and loiter times and, when needed, high speeds and accelerations.
Chapter 7   Engine in a Convective Air Stream

7.1   CFX Model Setup for Engine in a Convective Air Stream

The results from Chapter 6 are for the constant volume combustion jet engine operating in static conditions or when the engine is starting, idling or in slow movement on the ground. It is now important to determine the complete operating envelope of the engine in flight conditions, or when it is operating in a convective air stream.

The outer domain in CFX previously had an initial condition and boundary conditions of still air at a pressure of one bar and a temperature of 300K. The boundary condition at the outer domain inlet will now be modified for convective air at 100 m/s, 200 m/s, 225 m/s, 250 m/s and 275 m/s. These speeds correspond to Mach numbers at sea level standard conditions of 0.29, 0.58, 0.65, 0.72 and 0.80 respectively. Simulations at 300 m/s and higher were conducted, but resulted in quenching of the residual gases, thus reductions to the inlet open time or inlet area at high speeds may be required. This engine is quite possibly capable of higher speeds, including transonic and supersonic, but this study will only investigate up to 0.80 Mach. A depiction of the new boundary conditions relative to the engine is shown in Figure 7-1.
Pressure monitor points were placed in CFX just downstream of the new boundary condition (Freestream Pressure) and at a point on the inlet face of the compressor (Convective Air Pressure). Across the compressor inlet boundary condition, the incoming air pressure is increased per the compressor ratio, which in this study is a ratio of 2.5. An example of the convective air pressure monitors and the compressor operation is shown below in the CFX pressure depiction for 225 meters per second convective air and 8.56 grams per second fuel flow.

Figure 7-1: Outer Domain Inlet Boundary Condition for Convective Air Stream
7.2 Convective Air Simulations

Simulations were accomplished for each of the five chosen convective air speeds and at numerous fuel flow rates. Minimum fuel flow rates were noted when combustion was not sustained at a lower fuel flow due to an inadequate propane-air ratio. Fuel flow rates which corresponded to maximum attainable pressure and thrust were noted when peak pressure and thrust values stagnated when fuel flow was increased further. These peak fuel flow rates and thrust values correspond to the stoichiometric conditions for the respective airspeed, which is also evident by the corresponding peak temperatures around 2600K, the approximate
constant volume adiabatic flame temperature. CFX depictions of the temperatures and thrust for the 225 m/s and 8.56 grams/sec fuel flow are shown below.

**Figure: 7-3 Temperature and Thrust Plots for a 225 m/s and 8.56 grams/sec Convective Air Simulation (Wedge)**

CFX depictions of the pressures and thrust for the 275 m/s and 2.28, 3.71 and 12.13 grams/sec fuel flow are shown below.

**Figure 7-4: Pressure and Fuel Plots for a 275 m/s and 2.28, 3.71 and 12.13 gram/sec Propane Convective Air Simulation (Wedge)**
Figure 7-5: Convective Air at 275 m/s Sequential Temperature Images, Propane at 12.13 grams/sec

In the images above (Figure 7-5) are the sequential temperature images for the 275 m/s and 12.13 grams/sec engine fuel flow, which corresponds to the engine’s peak thrust operation of 191.6 Newtons, but at a specific impulse of only 1604 seconds. The image below (Figure 7-6) depicts the engine mach number during expansion of this same simulation. Of note is the
choked flow at the throat throughout the expansion and the peak nozzle exit plane mach number of 2.4. Also, overexpansion in the nozzle occurs late in the expansion, 600 timesteps prior to the inlet opening, or when 77% of the expansion time is complete.
Figure 7-6: Convective Air at 275 m/s Sequential Mach Number Images, 12.13 grams/sec Propane, Expansion Phase Only. Maximum Exhaust Mach Number is 2.4.
The following Tables 7-2 and 7-4 show the essential results from the convective air simulations. The average thrust and specific impulse results are then plotted as a function of the average fuel flow in Figures 7-7 and 7-8.
Table 7-1: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 100 m/s and 200 m/s

<table>
<thead>
<tr>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newtons)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>1.14</td>
<td>130</td>
<td>6.9</td>
<td>1650</td>
<td>44.1</td>
<td>3920.2</td>
</tr>
<tr>
<td>100</td>
<td>2.28</td>
<td>185</td>
<td>10.0</td>
<td>2065</td>
<td>62.7</td>
<td>2789.4</td>
</tr>
<tr>
<td>100</td>
<td>4.00</td>
<td>245</td>
<td>13.4</td>
<td>2300</td>
<td>83.1</td>
<td>2110.9</td>
</tr>
<tr>
<td>100</td>
<td>8.56</td>
<td>250</td>
<td>14.0</td>
<td>2300</td>
<td>84.8</td>
<td>1005.2</td>
</tr>
<tr>
<td>200</td>
<td>2.28</td>
<td>225</td>
<td>12.7</td>
<td>1910</td>
<td>76.3</td>
<td>3392.5</td>
</tr>
<tr>
<td>200</td>
<td>2.85</td>
<td>260</td>
<td>15.0</td>
<td>2035</td>
<td>88.2</td>
<td>3136.2</td>
</tr>
<tr>
<td>200</td>
<td>3.43</td>
<td>290</td>
<td>16.8</td>
<td>2140</td>
<td>98.4</td>
<td>2915.0</td>
</tr>
<tr>
<td>200</td>
<td>4.00</td>
<td>305</td>
<td>17.7</td>
<td>2110</td>
<td>103.4</td>
<td>2627.8</td>
</tr>
<tr>
<td>200</td>
<td>5.00</td>
<td>335</td>
<td>19.4</td>
<td>2510</td>
<td>113.6</td>
<td>2309.0</td>
</tr>
<tr>
<td>200</td>
<td>8.56</td>
<td>395</td>
<td>22.2</td>
<td>2675</td>
<td>134.0</td>
<td>1588.2</td>
</tr>
<tr>
<td>200</td>
<td>12.13</td>
<td>390</td>
<td>22.2</td>
<td>2380</td>
<td>132.3</td>
<td>1106.9</td>
</tr>
</tbody>
</table>
Table 7-2: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 225 m/s and 250 m/s

<table>
<thead>
<tr>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newton)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newton)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>225</td>
<td>228</td>
<td>245</td>
<td>13.2</td>
<td>1885</td>
<td>83.1</td>
<td>3694.0</td>
</tr>
<tr>
<td>225</td>
<td>314</td>
<td>290</td>
<td>16.9</td>
<td>2125</td>
<td>98.4</td>
<td>3180.0</td>
</tr>
<tr>
<td>225</td>
<td>400</td>
<td>330</td>
<td>18.5</td>
<td>2345</td>
<td>111.9</td>
<td>2843.2</td>
</tr>
<tr>
<td>225</td>
<td>500</td>
<td>365</td>
<td>21.0</td>
<td>2500</td>
<td>123.8</td>
<td>2515.8</td>
</tr>
<tr>
<td>225</td>
<td>856</td>
<td>425</td>
<td>24.2</td>
<td>2700</td>
<td>144.1</td>
<td>1708.8</td>
</tr>
<tr>
<td>225</td>
<td>12.13</td>
<td>425</td>
<td>24.6</td>
<td>2530</td>
<td>144.1</td>
<td>1206.2</td>
</tr>
<tr>
<td>250</td>
<td>228</td>
<td>285</td>
<td>13.6</td>
<td>1680</td>
<td>96.7</td>
<td>4297.1</td>
</tr>
<tr>
<td>250</td>
<td>428</td>
<td>370</td>
<td>21.3</td>
<td>2160</td>
<td>125.5</td>
<td>2975.3</td>
</tr>
<tr>
<td>250</td>
<td>542</td>
<td>410</td>
<td>23.6</td>
<td>2245</td>
<td>139.1</td>
<td>2602.9</td>
</tr>
<tr>
<td>250</td>
<td>614</td>
<td>425</td>
<td>25.1</td>
<td>2525</td>
<td>144.1</td>
<td>2384.9</td>
</tr>
<tr>
<td>250</td>
<td>856</td>
<td>475</td>
<td>27.8</td>
<td>2550</td>
<td>161.1</td>
<td>1909.8</td>
</tr>
<tr>
<td>250</td>
<td>12.13</td>
<td>510</td>
<td>29.2</td>
<td>2575</td>
<td>173.0</td>
<td>1447.5</td>
</tr>
<tr>
<td>250</td>
<td>17.84</td>
<td>510</td>
<td>27.8</td>
<td>2575</td>
<td>173.0</td>
<td>984.3</td>
</tr>
</tbody>
</table>
Table 7-3: Constant Volume Combustion Jet Engine Convective Air Simulation Results, 275 m/s

<table>
<thead>
<tr>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newtons)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>275</td>
<td>2.28</td>
<td>315</td>
<td>15.0</td>
<td>1355</td>
<td>106.8</td>
<td>4749.5</td>
</tr>
<tr>
<td>275</td>
<td>3.71</td>
<td>375</td>
<td>18.9</td>
<td>2005</td>
<td>127.2</td>
<td>3479.5</td>
</tr>
<tr>
<td>275</td>
<td>4.28</td>
<td>390</td>
<td>22.6</td>
<td>2010</td>
<td>132.3</td>
<td>3136.2</td>
</tr>
<tr>
<td>275</td>
<td>4.85</td>
<td>420</td>
<td>24.3</td>
<td>2020</td>
<td>142.4</td>
<td>2980.1</td>
</tr>
<tr>
<td>275</td>
<td>6.57</td>
<td>470</td>
<td>27.8</td>
<td>2280</td>
<td>159.4</td>
<td>2464.9</td>
</tr>
<tr>
<td>275</td>
<td>8.56</td>
<td>505</td>
<td>29.0</td>
<td>2640</td>
<td>171.3</td>
<td>2030.5</td>
</tr>
<tr>
<td>275</td>
<td>12.13</td>
<td>565</td>
<td>31.0</td>
<td>2640</td>
<td>191.6</td>
<td>1603.6</td>
</tr>
<tr>
<td>275</td>
<td>17.84</td>
<td>565</td>
<td>31.5</td>
<td>2600</td>
<td>191.6</td>
<td>1090.4</td>
</tr>
</tbody>
</table>
Figure 7-7: Average Engine Thrust vs. Average Engine Fuel Flow for Convective Stream of 100 m/s, 200 m/s, 225 m/s, 250 m/s and 275 m/s
Figure 7-8: Specific Impulse vs. Average Engine Fuel Flow for Convective Stream of 100 m/s, 200 m/s, 225 m/s, 250 m/s and 275 m/s
7.3 Conclusions

7.3.1 Performance of the constant volume combustion jet engine was dramatically improved in the convective air stream when compared to static conditions.

Peak thrust went from 45 Newtons at 2.28 grams/second of fuel, at static conditions, to 192 Newtons at 12.13 grams/second of fuel, while at 275 meters/second.

Peak specific impulse went from 3217 seconds at 0.86 grams/second, at static conditions, to 4750 seconds at 2.28 grams/second of fuel at 275 meters/second.

- 7.3.2 From the thrust vs. fuel flow chart, it is readily apparent where the engine stoichiometric fuel flow rates reside for each airspeed. Fuel flows above these values at those corresponding speeds should be avoided as fuel would be wasted.

7.3.3 The thrust turn-down ratio is 191.6/44.1 = 4.35. The fuel turn-down to achieve this thrust turn-down is 12.13/1.14 = 10
Chapter 8    Nozzle Optimization

8.1  Supersonic Nozzle Design

Exhaust nozzles of thrust engines are normally a convergent-divergent type, or de Laval nozzle. The flow through such a nozzle is usually axisymmetric and can be considered nearly one-dimensional or quasi-one-dimensional and it is also usually nearly isentropic. The exit-to-throat area ratio alone governs the exit-to-reservoir pressure ratio across the nozzle. For maximum thrust at a given pressure ratio, the pressure in the exit plane should theoretically be equal to the ambient pressure. The design complication for steady flow nozzles is usually due to the changing operating environment of the vehicle, i.e., the changing ambient pressure. This is most pronounced in vehicles with high altitude operating environments. Since a fixed nozzle can only be designed with one correct area and pressure ratio, there is only one altitude for which the gas can be properly expanded. At other altitudes, the gas may not be fully expanded at exit (an under-expanded condition), or it may be expanded below the ambient pressure (an over-expanded condition). A supersonic nozzle flow pattern summary is shown below in Figure 73. Either flow, under-expanded or over-expanded, produces losses in thrust efficiency, so it is customary to design the nozzle for some mean altitude (mean pressure ratio) so that it operates in an under-expanded condition at high altitude, and an over-expanded condition at sea level, where flow might separate. Changing operating altitudes are probably not a design factor for the constant volume combustion engine, as its operating envelope will most likely be limited to 5000 ft Mean Sea Level or less, where little changes to ambient pressure exist. However, these same design
considerations are of concern with the constant volume combustion jet engine due to the changing reservoir pressure during expansion.

In addition to the loss of nozzle thrust efficiency when operating at off design conditions, of serious concern is nozzle flow separation when the nozzle expansion ratio is too large for a given nozzle pressure ratio. When supersonic contour nozzle flow expands to a pressure level that is far lower than the ambient pressure, the formation of a normal shock occurs and then the detachment of the ensuing flow from the nozzle walls. This flow separation results from the flow in the boundary layer not having sufficient energy to overcome adverse pressure gradients. Because the flow separation often occurs on one side of the nozzle, causing azimuthal pressure distributions, side loads on the nozzle can cause serious structural concerns.

Unsteady nozzle flows cause the same design issues as those raised by the changing, off-design operating environments. Unsteady flows can result at engine start-up, shutdown, some other transient operations, or in some cases, it can result from the propulsion source itself, such as that found in a pulse engine thrust source. Unsteady flows and a changing ambient pressure coupled together, require a thorough understanding of the unsteady supersonic flow through nozzles. [12][13][20]

Obviously, the design concerns of this study are the unsteady flow through the engine. It is desired to find a nozzle design that eliminates or limits over-expansion, but also tries to minimize the inefficiencies of under-expansion.
Figure 8-1: Supersonic Nozzle Flow Conditions During Expansion [2]
The constant volume combustion engine of this study is an unsteady propulsion device, as the combustion chamber pressure is constantly changing during the expansion through the nozzle and also the expansion occurs for only 50% of each cycle. As stated above, a particular nozzle design on this engine will only operate at the design point at one time during the expansion – at other times the nozzle is usually in either an under-expanded or over-expanded state, but will actually see each one of the conditions shown in Figure 8-1 during each of the 194 cycles per second.

The constant volume combustion engine nozzle expansion is very similar to the design aspects of a rocket motor blowdown of the combustion chamber of a solid propellant rocket motor after the propellant is consumed. A typical pressure-time plot at the head of the grain for a rocket motor is shown below. [12]

![Figure 8-2: Rocket Motor Blowdown Similarity to Constant Volume Combustion Jet Engine Expansion](image)

Figure 8-2: Rocket Motor Blowdown Similarity to Constant Volume Combustion Jet Engine Expansion
For most of the rocket operating range, the mass flow rate of combustion gas through the perforation in the solid propellant grain changes very little with time. This portion of the flow is best analyzed as a quasi-steady flow analysis using a steady one-dimensional flow model with mass addition. During the ignition and blowdown phases, however, the flow properties change rapidly with time and the unsteady flow effects become significant. This blowdown phase is what is occurring in the constant volume combustion engine throughout the expansion through the nozzle.

The changing operating environment of this constant volume combustion jet engine is most likely not a design concern. This engine will normally operate between sea level and 5000 ft mean sea level, and thus close to standard temperature and pressure of 1 bar and 300K. These are the input values used in the CFX simulations as the ambient conditions in the “Outer” domain.

The objective of two-dimensional, steady-state supersonic nozzle design for a nozzle, as shown in the figure below, is to expand a gas at rest to a given supersonic Mach number at the exit, $M_e$. The desire is to design the area ratio and the contour to provide a shock-free, isentropic flow in the nozzle. The convergent, subsonic section has no specific required contour, only general rules of thumb for its design. The primary concern is the divergent supersonic portion. To begin the method of characteristics, an initial data line is used, which is downstream of the limiting characteristic, where the flow properties are known (see drawing). A mesh with grid points is built through the expansion section and the straightening section. The numerical solution of the flow is carried out by solving the compatibility equations along the characteristic lines in a step-by-step fashion, starting from
the initial data line. The flow properties at each grid point are determined using a finite-difference method, such as the Euler Predictor-Corrector. An illustration of such a finite-difference grid is shown below. [12] [16]

Figure 8-3: Method of Characteristics for Nozzle Design

8.2 CFX Nozzle Optimization Simulations

Nozzle designs in this study are being accomplished by using a publically available MATLAB nozzle design program called 2-D Nozzle Design. [Appendix D] This program gives the ideal nozzle geometry using the method of characteristics for a Quasi-2D Diverging Nozzle. It assumes that the gas that is exhausting from the combustion chamber has no mass
flow rate in. Using 2D nozzle flow relations, an optimal throat area is found that produces the maximum amount of thrust for the given ambient pressure and combustion chamber parameters. This exit area is automatically set and fed into the method of characteristics portion of that code. The method of characteristics also uses the exit Mach number that corresponds to the ideal exit area.

As an example of the output of this MATLAB code, shown below is the output for the 6 bar reservoir nozzle design used in all previous CFX simulations. [11]

**Figure 8-4: 2-D Nozzle Design Typical MATLAB Output – 6 bar Nozzle**

A cone-shaped nozzle, the simplest nozzle design, was used in all CFX simulations, using the nozzle area ratio and nozzle length from this code. This 2-D nozzle design code was computationally validated using CFX by expanding a nozzle designed for a 9 bar combustion
chamber reservoir pressure of propane-air products at a starting pressure \(10 \times 10^5 \text{ N/m}^2\) and temperature 2500K. Shown in Figure 8-5 below is a plot of the decreasing combustion chamber pressure, along with the points 1 – 4 that correspond to the images of engine pressure and mach number. Point No. 1 is an underexpanded condition where the 10 bar reservoir pressure is above the design pressure. Fully supersonic flow and any shocks are beyond the nozzle exit plane. Point 2 is at the design point of 9 bar. Flow is still supersonic beyond the exit plane with no shock waves. At Point 3, the reservoir pressure of 8 bar has dropped below the design pressure of 9 bar and the shock is now at the nozzle exit plane. Finally, at Point 4, the flow is now overexpanded, with the shock wave clearly evident at the nozzle midpoint. These are the expected results from expanding through this nozzle.

Nozzle designs were accomplished for combustion chamber reservoirs of 10 bar, 8 bar, 7 bar, in addition to the already simulated nozzle design of 6 bar. These nozzle designs are depicted below in Figures 8-6 and 8-7.
Figure 8-5: Validation of the 2-D Nozzle Design MATLAB Program
Previously, expansions through the current 6 bar nozzle design have not seen underexpanded conditions, except near the end of the expansion when thrust production has been completed, so there has been no adverse impact. The goal of the CFX simulations with four different nozzle designs is to find possible performance improvements in thrust production or higher specific impulse using a larger nozzle than the current 6 bar design. At some point when the nozzle area ratio is made larger, the overexpanded condition will adversely impact the thrust generation due to flow separation and nozzle side loads.
Numerous simulations were conducted using these nozzle designs at two convective air stream conditions – 225 meters per second and 275 meters per second – and at various fuel flow rates. These airsreads were selected because any adverse impact identified in the simulations from overexpansion at these higher compressor outlet conditions (and thus higher combustion chamber pressures) will be magnified with the same nozzle design and at lower convective airsreads (and lower combustion chamber pressures). The results from all of the simulations are shown below in Tables 8-1 and 8-2.

Figure 8-7: 8 bar and 10 bar Nozzle Design Geometries
Table 8-1: Results from 6 bar and 7 bar Nozzle Design Simulations

Nozzle Optimization Results - 6 and 7 Bar Reservoir Designs
Constant Volume Combustion Jet Engine with Convective Air Stream
(225 meters/second and 275 meters/second)

<table>
<thead>
<tr>
<th>Nozzle Reservoir Design</th>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newtons)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>3.14</td>
<td>290</td>
<td>16.9</td>
<td>2125</td>
<td>45.5</td>
<td>3180.0</td>
</tr>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>5.00</td>
<td>365</td>
<td>21.0</td>
<td>2500</td>
<td>123.8</td>
<td>2515.8</td>
</tr>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>8.56</td>
<td>425</td>
<td>24.2</td>
<td>2700</td>
<td>144.1</td>
<td>1708.8</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>3.71</td>
<td>375</td>
<td>18.9</td>
<td>2005</td>
<td>127.2</td>
<td>3479.5</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>6.57</td>
<td>470</td>
<td>27.8</td>
<td>2280</td>
<td>159.4</td>
<td>2464.9</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>12.13</td>
<td>565</td>
<td>31.0</td>
<td>2640</td>
<td>191.6</td>
<td>1603.6</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>2.28</td>
<td>225</td>
<td>13.2</td>
<td>1885</td>
<td>76.3</td>
<td>3392.5</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>3.14</td>
<td>300</td>
<td>16.9</td>
<td>2125</td>
<td>101.7</td>
<td>3289.7</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>5.00</td>
<td>375</td>
<td>21.0</td>
<td>2500</td>
<td>127.2</td>
<td>2584.7</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>8.56</td>
<td>430</td>
<td>24.6</td>
<td>2530</td>
<td>145.8</td>
<td>1728.9</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>3.14</td>
<td>275</td>
<td>15.0</td>
<td>1355</td>
<td>101.7</td>
<td>3289.7</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>3.76</td>
<td>340</td>
<td>19.0</td>
<td>2000</td>
<td>115.3</td>
<td>3154.7</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>6.57</td>
<td>485</td>
<td>27.8</td>
<td>2330</td>
<td>164.5</td>
<td>2543.5</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>12.13</td>
<td>555</td>
<td>31.0</td>
<td>2650</td>
<td>188.2</td>
<td>1575.2</td>
</tr>
</tbody>
</table>
Table 8-2: Results from 8 bar and 10 bar Nozzle Design Simulations

Nozzle Optimization Results - 8 and 10 Bar Reservoir Designs
Constant Volume Combustion Jet Engine with Convective Air Stream
(225 meters/second and 275 meters/second)

<table>
<thead>
<tr>
<th>Nozzle Reservoir Design</th>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Peak Thrust (Newtons)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>3.14</td>
<td>134</td>
<td>13.4</td>
<td>2125</td>
<td>45.5</td>
<td>1469.4</td>
</tr>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>5.00</td>
<td>358</td>
<td>18.9</td>
<td>2465</td>
<td>121.4</td>
<td>2467.6</td>
</tr>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>8.56</td>
<td>500</td>
<td>25.6</td>
<td>2745</td>
<td>169.6</td>
<td>2010.4</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>3.71</td>
<td>185</td>
<td>17.6</td>
<td>2320</td>
<td>62.7</td>
<td>1716.5</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>6.57</td>
<td>470</td>
<td>26.8</td>
<td>2350</td>
<td>159.4</td>
<td>2464.9</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>12.13</td>
<td>680</td>
<td>35.0</td>
<td>2765</td>
<td>230.6</td>
<td>1929.9</td>
</tr>
<tr>
<td>10 Bar</td>
<td>225</td>
<td>3.14</td>
<td>153</td>
<td>16.2</td>
<td>1778</td>
<td>51.9</td>
<td>1677.7</td>
</tr>
<tr>
<td>10 Bar</td>
<td>225</td>
<td>6.57</td>
<td>355</td>
<td>28.2</td>
<td>2545</td>
<td>120.4</td>
<td>1861.8</td>
</tr>
<tr>
<td>10 Bar</td>
<td>225</td>
<td>8.56</td>
<td>320</td>
<td>26.0</td>
<td>2210</td>
<td>108.5</td>
<td>1286.6</td>
</tr>
<tr>
<td>10 Bar</td>
<td>275</td>
<td>3.71</td>
<td>180</td>
<td>20.0</td>
<td>1570</td>
<td>61.1</td>
<td>1670.1</td>
</tr>
<tr>
<td>10 Bar</td>
<td>275</td>
<td>8.56</td>
<td>360</td>
<td>28.2</td>
<td>2545</td>
<td>122.1</td>
<td>1888.0</td>
</tr>
<tr>
<td>10 Bar</td>
<td>275</td>
<td>12.13</td>
<td>680</td>
<td>35.0</td>
<td>2765</td>
<td>230.6</td>
<td>1929.9</td>
</tr>
</tbody>
</table>
The resulting thrust and specific impulse values as a function of average fuel flow rates and convective air stream speeds are plotted below in Figures 8-8 thru 8-11.

Summarizing the results from the four nozzle designs at 225 m/s, the results for the 6 bar nozzle design were previously ascertained in Chapter 7 simulations with a convective airstream. All simulations with the 6 bar nozzle design at all modeled airspeeds and fuel flows showed no characteristics of flow separation from overexpansion, including at the minimum fuel flows. As is clear from Figures 8-8 thru 8-11, the 10 bar nozzle design suffered from severe overexpansion, with thrust values 20% – 40% less than the 6 bar nozzle design, and thus low specific impulses of about 1500 seconds. The 8 bar nozzle design showed a gain of approximately 20% in thrust at the stoichiometric fuel flow of 8.56 grams/second of propane, however specific impulse at this fuel flow was still only about 2000 seconds. At fuel flows below 5.0 grams/second, flow separation and overexpansion were clearly evident, as thrust and specific impulse dropped dramatically. Finally, the 7 bar nozzle design results were closely aligned with the 6 bar nozzle results except at the lowest fuel flow rate of 2.28 grams per second, where the 7 bar nozzle design thrust and specific impulse were both about 10% lower than the 6 bar nozzle design.

Reviewing the results from the four nozzle designs at 275 m/s, it is clear that the results are very similar to those of a 225 m/s convective airspeed and the performance reductions from the 6 bar nozzle results are just as evident.
Figure 8-8: 6, 7, 8 and 10 bar Nozzle Comparisons, Thrust vs. Fuel Flow, 225 m/s
Figure 8-9: 6, 7, 8 and 10 bar Nozzle Comparisons, Specific Impulse vs. Fuel Flow, 225 m/s
Figure 8-10: 6, 7, 8 and 10 bar Nozzle Comparisons, Thrust vs. Fuel Flow, 275 m/s
Figure 8-11: 6, 7, 8 and 10 bar Nozzle Comparisons, Specific Impulse vs. Fuel Flow, 275 m/s
8.3 Contoured Nozzle Analysis

The cone-shaped nozzles analyzed in all previous simulations have worked well in producing thrust and are simple nozzles that are ideal for a simple device like this constant volume combustion engine. However, it is worthwhile to investigate whether contouring the nozzle into smooth transitions from the engine throat to the nozzle exit plane. It is possible that a nozzle contour may permit the nozzles to increase thrust and possibly reduce or eliminate flow separation while operating in the low fuel flow and low combustion chamber operating regions.

The contoured nozzles for the 6, 7 and 8 bar designs are depicted in Figures 8-12 thru 8-14 below. These nozzles maintain the same area ratios and lengths as previously used for the cone-shaped nozzles. Simulations were accomplished for convective airspeeds of 225 m/s and 275 m/s for each of these contoured nozzles. The results are shown in Table 8-3.

![Fig. 8-12: 6 bar Contoured Nozzle](image1)

![Fig. 8-13: 7 bar Contoured Nozzle](image2)

![Fig. 8-14: 8 bar Contoured Nozzles](image3)
Table 8-3: Simulation Results for 6, 7 and 8 bar Contoured Nozzles at 225 m/s and 275 m/s

<table>
<thead>
<tr>
<th>Contoured Nozzle Design</th>
<th>Convective Stream Velocity (meters/sec)</th>
<th>Average Total Engine Fuel Flow (g/min/sec)</th>
<th>Peak Thrust (Newton)</th>
<th>Peak Combustion Chamber Pressure (Bar)</th>
<th>Peak Combustion Chamber Temperature (Kelvin)</th>
<th>Average Thrust (Newton)</th>
<th>Specific Impulse (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>2.28</td>
<td>225</td>
<td>13.2</td>
<td>1885</td>
<td>58.5</td>
<td>2617.1</td>
</tr>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>3.14</td>
<td>380</td>
<td>16.9</td>
<td>2125</td>
<td>81.4</td>
<td>2631.7</td>
</tr>
<tr>
<td>6 Bar</td>
<td>225</td>
<td>8.56</td>
<td>385</td>
<td>24.2</td>
<td>2700</td>
<td>123.1</td>
<td>1459.5</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>2.28</td>
<td>380</td>
<td>15.0</td>
<td>1355</td>
<td>86.8</td>
<td>3859.9</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>3.71</td>
<td>315</td>
<td>18.5</td>
<td>2005</td>
<td>91.6</td>
<td>2503.2</td>
</tr>
<tr>
<td>6 Bar</td>
<td>275</td>
<td>12.13</td>
<td>500</td>
<td>31.0</td>
<td>2640</td>
<td>159.9</td>
<td>1338.0</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>2.28</td>
<td>240</td>
<td>13.2</td>
<td>1885</td>
<td>46.5</td>
<td>2067.8</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>3.14</td>
<td>300</td>
<td>16.9</td>
<td>2125</td>
<td>81.4</td>
<td>2631.7</td>
</tr>
<tr>
<td>7 Bar</td>
<td>225</td>
<td>8.56</td>
<td>360</td>
<td>24.2</td>
<td>2700</td>
<td>111.6</td>
<td>1323.4</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>2.28</td>
<td>280</td>
<td>15.0</td>
<td>1355</td>
<td>67.8</td>
<td>3015.5</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>3.71</td>
<td>325</td>
<td>18.5</td>
<td>2005</td>
<td>88.2</td>
<td>2412.4</td>
</tr>
<tr>
<td>7 Bar</td>
<td>275</td>
<td>12.13</td>
<td>450</td>
<td>31.0</td>
<td>2640</td>
<td>148.3</td>
<td>1240.7</td>
</tr>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>2.28</td>
<td>320</td>
<td>13.2</td>
<td>1885</td>
<td>37.2</td>
<td>1654.2</td>
</tr>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>3.14</td>
<td>325</td>
<td>16.9</td>
<td>2125</td>
<td>85.0</td>
<td>2749.2</td>
</tr>
<tr>
<td>8 Bar</td>
<td>225</td>
<td>8.56</td>
<td>380</td>
<td>24.2</td>
<td>2700</td>
<td>117.8</td>
<td>1396.9</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>2.28</td>
<td>350</td>
<td>15.0</td>
<td>1355</td>
<td>40.7</td>
<td>1809.3</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>3.71</td>
<td>380</td>
<td>18.5</td>
<td>2005</td>
<td>95.7</td>
<td>2619.2</td>
</tr>
<tr>
<td>8 Bar</td>
<td>275</td>
<td>12.13</td>
<td>450</td>
<td>31.0</td>
<td>2640</td>
<td>139.5</td>
<td>1167.7</td>
</tr>
</tbody>
</table>
The contoured nozzle simulation results for thrust and specific impulse are plotted in Figures 8-15 thru 8-16, where they are compared to the 6 bar cone-shaped nozzle.

![Graph showing thrust comparison](image)

Fig. 8-15: 6, 7 and 8 bar Contoured Nozzle Thrust Comparison to 6 bar Cone-Shaped Nozzle at 225 m/s
Fig. 8-16: 6, 7 and 8 bar Contoured Nozzle Specific Impulse Comparison to 6 bar Cone-Shaped Nozzle at 225 m/s

Fig. 8-17: 6, 7 and 8 bar Contoured Nozzle Thrust Comparison to 6 bar Cone-Shaped Nozzle at 275 m/s
It is clear from the above results that the use of contoured nozzles for this engine is resulting in less total thrust and additional flow separation, especially at low reservoir pressures.
8.4 Conclusions

As was stated in Chapter 1, the driving design factor for this engine is specific impulse, since the design with the highest specific impulse can then be scaled up to meet the thrust design requirements, without changing the specific impulse. It is very clear that the 6 bar cone-shaped nozzle design is an optimum geometry, since any larger nozzle area ratio reduces performance through flow separations of overexpansion at the lower fuel flow rates where high specific impulse occurs. And any smaller nozzle area ratio simply leaves the gas even more underexpanded at the higher flow pressures, which will also reduce performance.
Chapter 9  Thermodynamic Analysis and Thrust-to-Weight Ratios

9.1  Thermodynamic Analysis

A First Law of Thermodynamics analysis of the constant volume combustion jet engine will now be completed, which will provide overall cycle thermal efficiency and overall engine efficiency, which can then be compared to other propulsion engine types. A pressure-volume (P-V) diagram and a temperature-entropy diagram (T-s) are shown below in Figure 9-1, with a summary of the state points to be used in the analysis. Pressure and temperatures are gathered from monitor points chosen to be located in the CFX model during the simulations for output to the results file. Enthalpies and entropies are then manually determined from the respective tables. [16][22]

![Humphrey Cycle Pressure-Volume (P-V) and Temperature-Entropy (T-s) Diagrams](image)

*Figure 9-1: Humphrey Cycle Pressure-Volume (P-V) and Temperature-Entropy (T-s) Diagrams*
1 – 2  Isentropic Compression, Compressor and Convective Air
1’ - 2  Non-Isentropic Compression, Compressor with Losses
2 – 3  Isochoric Heat Addition, Combustion Chamber
3 – 4  Isentropic Expansion, Nozzle
3 – 4’ Non-Isentropic Expansion, Nozzle with Losses
4 – 1  Isobaric Heat Rejection, Nozzle with Open Air Inlet

State 1 pressure and temperature are known from a monitor point located in the CFX model at the inlet face of the compressor. State 2 pressure and temperature are also determined from a monitor point – this one is located at the compressor outlet face and the pressure here is nominally set to be 2.5 times larger than the State 1 pressure. The State 1 and 2 values are relatively constant throughout each cycle for a given convective airspeed.

There are two pressure and temperature monitor points located in the combustion chamber, one set at the forward end and one set at the rearward end of the chamber. State 3 conditions are determined by averaging the two monitor points’ peak values for pressure and temperature. Enthalpy and entropy are then determined from the tables.

State 4’ is found from two monitor points at the nozzle exit plane for pressure and temperature. These values are recorded at the time just prior to the air inlet valve opening, when the expansion process is complete. Enthalpy and entropy are determined from these values.

State 4 is the point to which isentropic expansion would result when expanded to the same
pressure as State 4’. Enthalpies and entropies at these two points are used to determine the nozzle efficiency and the State 4 temperature.

\[ \eta_{\text{nozzle}} = \frac{\text{actual expansion enthalpy change}}{\text{isentropic expansion enthalpy change}} \]
\[ = \frac{(h_3 - h_{4'})}{(h_3 - h_4)} \]

Finally, State 1’ is the point from which the compressor must begin compression of the incoming air when applying a stated nozzle efficiency in order to meet the required conditions at the compressor outlet. For this model, a typical compressor efficiency of 70% will be applied.

\[ h_{1'} = h_2 - \frac{(h_2 - h_1)}{\eta_{\text{compressor}}} \]

The engine’s thermal cycle efficiency is defined as

\[ \eta_{\text{cycle}} = \frac{\text{Cycle Work Out}}{\left[ \text{Heat Added} + \text{Work Added} \right]} \]
\[ = \frac{[h_3 - h_{1'}]}{[h_3 - h_2 + h_2 - h_{1'}]} \]

Normally for this cycle, as depicted in the diagrams above, the only work added to the cycle is the compressor work. However, two additional work elements must be considered to get an accurate picture of the overall engine efficiency. First, this engine’s compressor will operate continuously, and since this engine is a pulse engine, the thermal cycle efficiency as
defined above does not account for all compressor work. Thus, it is necessary to find the engine efficiency, defined as

\[
\eta_{\text{engine}} = \frac{[h_3 - h_4']}{[h_3 - h_2] + 5[h_2 - h_1'] + W_{\text{Valves}}}
\]

where the compressor worked added has been multiplied by five to account for the unused but operating compressor work – the inlet is open 20% of the cycle and it is closed for 80% of the cycle.

In addition to this compensation for a constantly operating compressor, the operation of the inlet and throat valves must be included in the total engine work into the system. This work will most likely come in the form of a miniaturized, direct drive, electrically operated synchronous motor. The motor is best powered through batteries, and an estimate of the input work is necessary to determine the engine’s overall efficiency.

Research into small, dc-powered motors which might fit the application has found that power requirements for a 6000 rpm, 12-volt DC, high-torque motor are less than 50 watts. An example of one such motor is outlined in the Appendix. Converting these power requirements and allowing for two motors or a larger motor (100 Joules/sec), if necessary, into a compatible form useful for this analysis, i.e., Joules per gram of fuel-air mass flow

\[
\text{Power}_{\text{Valves}} = 100 \text{ watts} = 100 \text{ Joules/sec}
\]

Converting to a work per unit of fuel-air mass flow basis
\[ \text{Work}_{\text{Valves}} = \frac{\text{Power}_{\text{Valves}}}{\text{Engine Average Fuel-Air Mass Flow Rate}} \]

The approximate engine average fuel-air mass flow rate varies between 21 and 220 grams/second, depending on the inlet air compression, or a range of

\[ W_{\text{Valves}} = \frac{(100 \text{ J/sec})}{(21 \text{ grams/second})} = 4.76 \text{ KJ/kg} \]

up to

\[ W_{\text{Valves}} = \frac{(100 \text{ J/sec})}{(220 \text{ grams/second})} = 0.45 \text{ KJ/kg} \]

Note that the work input from the motor to drive the valves is a small portion of system work input, with the electric motor never requiring more than 2% of the compressor work requirement.

The complete analysis is detailed in the table located in Appendix E. A summary of the results is shown in Table 9-1 below. The average cycle thermal efficiency for the cases reviewed - maximum specific impulse and maximum thrust cases at all studied convective airspeeds - was 49%. When including the compressor work, when the compressor is operating but not contributing to the cycle, and the work of the valves’ drive motor, this engine efficiency is 37%. As a comparison, the average efficiency when assuming constant specific heat is 51%. Note that the decreases in overall engine efficiency are more
significant for the high specific impulse cases than for the high thrust cases. This is to be expected, since the efficiencies are based upon the total rise in enthalpy during the combustion phase, and the high I<sub>sp</sub> cases have much lower peak pressures, temperatures, and enthalpies. Thus, any increases in enthalpy as work into the engine will have a more negative effect on these efficiency calculations. Still, high specific impulse is the desired characteristic of this engine and this should be considered when reviewing the efficiency values.
Table 9-1: Constant Volume Combustion Jet Engine Thermodynamic Analysis Summary

**Constant Volume Combustion Jet Engine Thermodynamic Analysis**

<table>
<thead>
<tr>
<th>Convective Stream Velocity and Case Type</th>
<th>Average Total Engine Fuel Flow (grams/sec)</th>
<th>Average Thrust (Newtons)</th>
<th>Specific Impulse (seconds)</th>
<th>Nozzle Efficiency</th>
<th>Thermal Cycle Efficiency</th>
<th>Engine Efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>275 m/s Max isp</td>
<td>2.28</td>
<td>106.8</td>
<td>4750</td>
<td>0.85</td>
<td>0.53</td>
<td>0.36</td>
</tr>
<tr>
<td>275 m/s Max Thrust</td>
<td>12.13</td>
<td>191.6</td>
<td>1604</td>
<td>0.82</td>
<td>0.51</td>
<td>0.43</td>
</tr>
<tr>
<td>250 m/s Max isp</td>
<td>2.28</td>
<td>96.7</td>
<td>4297</td>
<td>0.89</td>
<td>0.51</td>
<td>0.37</td>
</tr>
<tr>
<td>250 m/s Max Thrust</td>
<td>12.13</td>
<td>173.0</td>
<td>1448</td>
<td>0.94</td>
<td>0.58</td>
<td>0.48</td>
</tr>
<tr>
<td>225 m/s Max isp</td>
<td>2.28</td>
<td>83.1</td>
<td>3694</td>
<td>0.88</td>
<td>0.48</td>
<td>0.37</td>
</tr>
<tr>
<td>225 m/s Max Thrust</td>
<td>8.56</td>
<td>144.1</td>
<td>1709</td>
<td>0.95</td>
<td>0.56</td>
<td>0.47</td>
</tr>
<tr>
<td>200 m/s Max isp</td>
<td>2.28</td>
<td>76.3</td>
<td>3393</td>
<td>0.91</td>
<td>0.49</td>
<td>0.40</td>
</tr>
<tr>
<td>200 m/s Max Thrust</td>
<td>8.56</td>
<td>134.0</td>
<td>1588</td>
<td>0.90</td>
<td>0.53</td>
<td>0.46</td>
</tr>
<tr>
<td>100 m/s Max isp</td>
<td>1.14</td>
<td>44.1</td>
<td>3920</td>
<td>0.86</td>
<td>0.39</td>
<td>0.31</td>
</tr>
<tr>
<td>100 m/s Max Thrust</td>
<td>8.56</td>
<td>84.8</td>
<td>1005</td>
<td>0.94</td>
<td>0.50</td>
<td>0.43</td>
</tr>
<tr>
<td>Static Max isp</td>
<td>0.86</td>
<td>24.4</td>
<td>3217</td>
<td>0.89</td>
<td>0.39</td>
<td>0.31</td>
</tr>
<tr>
<td>Static Max Thrust</td>
<td>2.28</td>
<td>44.8</td>
<td>1990</td>
<td>0.91</td>
<td>0.42</td>
<td>0.36</td>
</tr>
</tbody>
</table>

Average Thermal Efficiency = 49% 
Average Engine Efficiency = 37%
9.2 Thrust-to-Weight Ratios

It is now desired to determine an important engine performance parameter, the thrust-to-weight ratios, which might shed light on the engine’s acceleration, payload capacity, range, maximum speed, and loiter time. First, an estimate of the weight of the engine and engine accessories will be made.

The constant volume combustion jet engine will be subjected to extreme pressures and temperatures, up to 35 bar and 3000K. The material and thickness of the pulsejet, which was subjected to no more than 3 bar and 1800K, will not suffice, of course. One method of approximating the mass of the engine is to treat it as a stainless steel pressure vessel and use a quick estimate calculation commonly used in pressure vessel design. [14] The estimated mass of a cylindrically shaped pressure vessel with hemispherical ends is

\[ M_{\text{Engine}} = 2 \pi R^2 (R+L)P \left( \frac{\rho}{\sigma} \right) \]

where \( R \) = radius, \( P \) = gage pressure, \( L \) = length, \( \rho \) = material density, \( \sigma \) = material maximum working stress. Substituting,

\[
\begin{align*}
R &= 1.2878 \text{ inches} = 0.0327 \text{ meter} \\
P &= 35 \times 10^5 \text{ N/m}^2 = 35 \text{ bar} \\
L &= 8.25 \text{ inches} = 0.2096 \text{ meters} \\
\rho &= 8000 \text{ kg/m}^3 \text{ for steel} \\
\sigma &= \sigma_{\text{yield point}}/N \text{ where } N= \text{Safety Factor} = 4 \\
&= 400 \text{ MPa}/4 = 100 \text{ Mpa} = 100 \times 10^6 \text{ N/m}^2
\end{align*}
\]
which gives approximate mass of the engine as

\[ M_{\text{Engine}} = 0.4558 \text{ kg} = 1.005 \text{ pounds} \]

Since the engine surface area is approximately 55 in\(^2\) and stainless steel density is 0.289 lb/in\(^3\), the wall thickness will be

\[
\text{Engine Wall Thickness} = \frac{(1.005 \text{ lb})}{(55 \text{ in}^2)(0.289 \text{ lb/in}^3)} = 0.06323 \text{ in}
\]

or approximately 1/16 inch thick stainless steel wall.

Additionally, the inlet and throat valves are estimated at total weight of 1.8 pounds.

An estimate of the electric motor weight can be found in the specifications shown in Appendix F for the small 6000 rpm motor. This typical motor weight is 0.275 kg or 0.606 pounds.

Two possible types of batteries for the electric motor for the inlet and throat valves are shown in Appendix G. A 12-volt lithium-ion battery holding 30 minutes of charge for the selected motor weighs approximately 0.33 pounds. A Ni-MH battery holding approximately 30 minutes of charge weighs approximately 0.5 pounds.

The compressor batteries are estimated by:

- Compressor Work = 140 KJ/kg

- Air Mass Flow = 50 grams/sec (225 m/s, High Isp)

- Compressor Power = \((140 \text{ J/g})(50 \text{ g/sec}) = 7000 \text{ J/sec} = 7 \text{ kW}\)

- For 15 minutes, battery storage = 1.75 kW-hr = 1750 W-h

- Battery Weight = 1 pound per 100 W-h

- Total Battery Weight = 17.5 pounds

The compressor is estimated at a total of 1.0 pounds.
Finally, the propane storage tank, a small cylindrical pressure vessel, should hold at least 30 minutes of fuel supply. Using a propane flow rate of 2.28 grams/second, which is the maximum specific impulse flow rate, 2.0 kg or approximately 4 pounds of propane would need to be stored. Also, the propane storage tank at 125 psig, will be approximately 0.5 pounds.

The major propulsion component weights and the calculated thrust-to weight-ratios at common flight speeds are shown in Table 9-2 below.
Table 9-2: Thrust-to-Weight Ratio Summary for Constant Volume Combustion Jet Engine

<table>
<thead>
<tr>
<th>Engine Component</th>
<th>Specifications</th>
<th>Weight</th>
<th>Thrust-to-Weight Ratio Static Maximum Thrust</th>
<th>Thrust-to-Weight Ratio at 225 m/s Maximum Thrust</th>
<th>Thrust-to-Weight Ratio at 225 m/s and Maximum Isp</th>
<th>Thrust-to-Weight Ratio at 275 m/s and Maximum Thrust</th>
<th>Thrust-to-Weight Ratio at 275 m/s and Maximum Isp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine</td>
<td>8.25 inches long, 1.29 inches maximum radius</td>
<td>1.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inlet and Throat Valves</td>
<td>2.5 in radius and 0.6 in radius with center holes</td>
<td>1.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Motor, Synchronous, Electric</td>
<td>6000 rpm, 12-volt, 40 watt input</td>
<td>0.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Battery, Lithium Ion</td>
<td>1750 Amp. 19.4 watt-hour</td>
<td>0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Valve Drive Mechanism</td>
<td>Linkage Between Motor, Inlet and Throat Valve</td>
<td>0.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Compressor and Battery</td>
<td>Compressor Ratio 2.5 Lithium Ion Battery</td>
<td>1.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total Engine Component</td>
<td></td>
<td>22.03</td>
<td>0.5</td>
<td>1.5</td>
<td>0.9</td>
<td>2.0</td>
<td>1.1</td>
</tr>
<tr>
<td>Liquified Propane</td>
<td>30 minute supply at 2.28 grams/sec, 4 kg total</td>
<td>8.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Propane Storage Tank</td>
<td>4 kg total propane at 125 psig</td>
<td>0.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total Engine and Fuel</td>
<td></td>
<td>30.53</td>
<td>0.3</td>
<td>1.1</td>
<td>0.6</td>
<td>1.4</td>
<td>0.8</td>
</tr>
<tr>
<td>Aircraft and Payload</td>
<td>Assume 3 pound aircraft/1.0 pound payload</td>
<td>4.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total</td>
<td></td>
<td>34.53</td>
<td>0.3</td>
<td>0.9</td>
<td>0.5</td>
<td>1.3</td>
<td>0.7</td>
</tr>
</tbody>
</table>

*Saturday, November 23, 13*
For comparison purposes, here are some common engine and aircraft thrust-to-weight ratios [14]:

<table>
<thead>
<tr>
<th>Engine or Aircraft</th>
<th>Thrust-to-Weight Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pratt and Whitney F119</td>
<td>7.95</td>
</tr>
<tr>
<td>Rolls-Royce 593</td>
<td>5.4</td>
</tr>
<tr>
<td>J58 Jet Engine (SR-71)</td>
<td>5.2</td>
</tr>
<tr>
<td><strong>Constant Volume Combustion Jet Engine (Engine and Batteries)</strong></td>
<td><strong>2.0</strong></td>
</tr>
<tr>
<td><strong>Constant Volume Combustion Jet Engine, w/30 Minutes of Fuel</strong></td>
<td><strong>1.4</strong></td>
</tr>
<tr>
<td><strong>Constant Volume Combustion Aircraft, w/Fuel &amp; Payload</strong></td>
<td><strong>1.3</strong></td>
</tr>
<tr>
<td>F-16</td>
<td>1.1</td>
</tr>
<tr>
<td>MIG-29</td>
<td>1.09</td>
</tr>
<tr>
<td>F-15</td>
<td>1.04</td>
</tr>
<tr>
<td>Concorde</td>
<td>0.373</td>
</tr>
</tbody>
</table>

9.3 Conclusions

9.3.1 The thermodynamic cycle efficiency of the constant volume combustion jet engine is as good, if not better than most other types of engine cycles, averaging 49% when averaged across the wide range of fuel flow rates and airspeeds. The compressor work input is relatively constant across the convective airspeeds and the valve motor work requirement is a small percentage of the total work input into the system. When accounting for the
compressor work input when the compressor is operating with the inlet closed, the engine average efficiency drops to 37%. Cycle and engine efficiency do not directly include thrust in their values. Specific impulse, the measure of thrust per unit of fuel, is the most important characteristic to use in determining this engine’s usefulness, and from results shown earlier, this engine has specific impulse values as high as 4800 seconds, which compares extremely well with similar propulsion types.

![Figure 9-2: Comparison of Constant Volume Combustion Jet Engine with Other Propulsion Sources – Specific Impulse vs. Mach Number](image)

*Figure 9-2: Comparison of Constant Volume Combustion Jet Engine with Other Propulsion Sources – Specific Impulse vs. Mach Number*
9.3.2 The thrust-to-weight ratio calculations made some necessary assumptions and estimates to arrive at total engine and aircraft package weights. The thrust-to-weight ratios for the entire aircraft, engine, batteries, fuel and payload, range from 0.5 to 2.0 in flight at higher airspeeds and approximately, 0.3 at static, on the ground conditions. These are good values for a fully loaded vehicle and it is important to note that they include batteries and fuel storage, the two largest single weights, in their values. These thrust-to-weight ratios are high enough that flexibility is available to increase weights such as a higher payload, sturdier engine structure, more fuel, or other desired vehicle or engine components.
Chapter 10  Study Findings and Future Work

10.1  Study Findings

The major findings of this study include:

10.1.1  For the original engine geometry, the optimum cycle time is $5.15 \times 10^{-3}$ seconds or 194hz.

10.1.2  The optimum inlet open time (flushing and recharge) is 900 timesteps, or $0.9 \times 10^{-3}$ seconds.

10.1.3  A cycle expansion time of 2600 timesteps, or $2 \times 10^{-3}$ seconds and a compression time of 1550 timesteps, or $1.55 \times 10^{-3}$ seconds is desired for maximum thrust.

10.1.4  Regarding the five initial geometries and five simulated frequencies, the best performance occurs with the single inlet, reduced combustion chamber and reduced throat model, operating at 194hz. At this operating point, average thrust is 51 Newtons, average fuel flow is 1.92 grams per second, and specific impulse is 2700 seconds.

10.1.5  There are no distinct advantages from using a dual fuel injection scheme or a single fuel injection scheme with initial fuel velocities, especially when considering the added engine complexities inherent in those cases. Therefore the single fuel injection in the inlet airstream, with no initial velocity should be used. It is also evident that closing the throat valve and inlet at the same time should be modeled in future simulations to aid in reducing fuel losses.

10.1.6  There was no significant reduction in performance when comparing the single injection fuel mixing model with the premixed fuel-air model.
10.1.7 The premixed fuel-air model at 194 hz produced average thrust of 51 Newtons, at an average fuel flow of 1.92 grams per second, and had a specific impulse of 2700 seconds.

10.1.8 The single injection fuel mixing model produced 54 Newtons of thrust at a specific impulse of 2700 seconds. Also, a maximum thrust of 62 Newtons was produced at the minimum specific impulse of 2100 seconds and a thrust of 31 Newtons occurred at the maximum specific impulse value of 3200 seconds.

10.1.9 Peak thrust when the valves are modeled using a sinusoidal mass flow profile occurs at 2.28 grams per second at approximately 45 Newtons of thrust. Comparing this to the peak thrust obtained with binary valves, modeled in Chapter 5, which occurred at 3 grams per second and approximately 62 Newtons, there is a 28% reduction in peak thrust production when using the globe valves. However, the relocation of the air inlet to the longitudinal axis and simultaneous closing of the inlet and throat valves, together improved fuel-air mixing, reduced fuel losses through the throat valve, and allowed the peak thrust to be obtained at 24% less average fuel flow. This results in maintaining a peak specific impulse of over 3200 seconds with the sinusoidal mass flow profiles, or globe valves.

10.1.10 From the thrust and specific impulse plots, a rewarding find of the simulations is the outstanding turn-down capability of the engine - approximately 5:1. Peak thrust occurs at 2.28 grams per second and peak specific impulse occurs at 0.43 grams per second. This provides a propulsion system capable of providing long range and loiter times and, when needed, high speeds and accelerations.

10.1.11 Performance of the constant volume combustion jet engine was dramatically improved in the convective air stream when compared to static conditions.
Peak thrust went from 45 Newtons at 2.28 grams/second of fuel, at static conditions, to 192 Newtons at 12.13 grams/second of fuel, while at 275 meters/second.

Peak specific impulse went from 3217 seconds at 0.86 grams/second, at static conditions, to 4750 seconds at 2.28 grams/second of fuel at 275 meters/second.

10.1.12 From the convective air stream simulations’ thrust vs. fuel flow chart, it is readily apparent where the engine stoichiometric fuel flow rates reside for each airspeed. Fuel flows above these values at those corresponding speeds should be avoided as fuel would be wasted.

10.1.13 In a convective air stream, the thrust turn-down ratio is 191.6/44.1 = 4.35. The fuel turn-down to achieve this thrust turn-down is 12.13/1.14 = 10.

10.1.14 From the nozzle optimization analysis, it is very clear that the 6 bar cone-shaped nozzle design is an optimum geometry, since any larger nozzle area ratio reduces performance through flow separations and overexpansion at the lower fuel flow rates where high specific impulse occurs. And any smaller nozzle area ratio simply leaves the gas even more underexpanded at the higher flow pressures, which will also reduce performance. Contouring the shape of the diverging portion of the nozzle does not appear to be advantageous.

10.1.15 The thermodynamic cycle efficiency of the constant volume combustion jet engine is as good, if not better than most other types of engine cycles, averaging 49% when averaged across the wide range of fuel flow rates and airspeeds. The compressor work input is relatively constant across the convective airs speeds and the valve motor work requirement is a small percentage of the total work input into the system. When accounting for the compressor work input when the compressor is operating with the inlet closed, the engine
average efficiency drops to 37%. Cycle and engine efficiency do not directly include thrust in their values. Specific impulse, the measure of thrust per unit of fuel, is the most important characteristic to use in determining this engine’s usefulness, and from results shown earlier, this engine has specific impulse values as high as 4800 seconds, which compares extremely well with similar propulsion types.

10.1.16 The thrust-to-weight ratios for the entire aircraft, engine, batteries, fuel and payload, range from 0.5 to 2.0 in flight at higher airspeeds and approximately, 0.3 at static, on the ground conditions. These are good values for a fully loaded vehicle and it is important to note that they include batteries and fuel storage, the two largest single weights, in their values.
10.2 Future Work

The following is a list of future work to accomplish to further develop the constant volume combustion jet engine:

10.2.1 Continue to investigate nozzle improvements through contouring of the nozzle to prevent flow separation in a larger nozzle, which may give an increase in thrust for given fuel flow rates, thus increasing specific impulse.

![Possible Nozzle Contour Improvements](image)

Figure 10-1: Possible Nozzle Contour Improvements

10.2.2 Investigate engine performance in transonic and supersonic convective air streams and develop techniques, such as gradually reducing the inlet area, to promote continued combustion at these higher airspeeds.

10.2.3 Investigate other possible fuel sources.

10.2.4 Investigate alternate compressor power sources to lower engine weight and raise thrust-to-weight ratios.

10.2.5 Construct a working model of this engine.
REFERENCES


3. DARPA Vulcan Industry Day Briefing, Dr. Tom Bussing, June 8, 2008.


Appendix A - Initial CFX Engine Geometry
Appendix B – CFX–Pre User Expressions for Control and Analysis

LIBRARY:

CEL:

EXPRESSIONS:

Densityin = areaAve(Density)@inletarea Side 2
Densityout = areaAve(density)@outletarea Side 1 1
FRC = (force_z())@outer Default+force_z()@combustion chamber Default \ +force_z()@valvein Default +areaAve(p)@REGION:F1664.1450 \ +area()@REGION:F1664.1450-areaAve(p)@REGION:F1383.1454)
*area()@outletarea Side 2 +area()@inletarea Side 2 \ +(areaAve(p)@REGION:F1410.1418-areaAve(p)@REGION:F1383.1454) \ *area()@REGION:F1383.1454)*45
P3 = probe(p)@p3
P5 = areaAve(p)@outletarea Side 1 1
Pconv = areaAve(p)@super1
Pconvstream = probe(p)@C3H8exhaust
Pin = min(Pset*2.5, Psensor2*2.5)
Psensor = probe(Pressure)@p1
Psensor2 = areaAve(p)@super2
Pset = 1e5[Pa]
T0 = 293[K]
Thrustout = Densityout*vout*area(vout)*area()@outletarea Side 2 1 *45
Vortex Fuel = (0.0[kg m^-1s^-1]*(1/(lossin+1 [m^-1])))
componz = step(x/x0+7.4)-step(x/x0+2.4)
conductivity = Dynamic Viscosity*Cp/0.72
coscycle = cos(2.0*pi*(t+750*tstep)/(tstep*5160))
coscycle10 = \ ((-0.5*(((sin(1.0*pi*(t-4360*tstep)/(tstep*5160))))+abs(((sin(1.0*pi\ (t-4360*tstep)/(tstep*5160))))))-0.5*(((sin(1.0*pi*(t-4360*tstep)/(t\ step*5160))))+abs(((sin(1.0*pi*(t-4360*tstep)/(tstep*5160))))))+2)/2
coscycle2 = 2*abs((sin(2.0*pi*(t-1450*tstep)/(tstep*5160)))-1)
coscycle4 = \ -0.5*(((sin(4.0*pi*(t-2100*tstep)/(tstep*5160))))+abs(((sin(4.0*pi/\t\
\[ \text{step}(\pi(t-4360)*\text{tstep}/(\text{tstep}\cdot5160))) + \text{abs}(\sin(1.0\pi(t-1450)*\text{tstep}/(\text{tstep}\cdot5160))) + 2)/2 \]
\[ \text{coscycle2} = \text{step}(\sin(2.0\pi(t-1450)*\text{tstep}/(\text{tstep}\cdot5160))) - 1 \]
\[ \text{coscycle4} = \text{step}(\text{abs}(\sin(4.0\pi(t-2100)*\text{tstep}/(\text{tstep}\cdot5160))) - 1) \]
\[ \text{coscycle5} = \text{step}(\text{abs}(\sin(4.0\pi(t-2100)*\text{tstep}/(\text{tstep}\cdot5160))) + 1) \]
\[ \text{coscycle6} = \text{step}(\text{abs}(\sin(4.0\pi(t-2100)*\text{tstep}/(\text{tstep}\cdot5160))) - 1) \]
\[ \text{coscycle9} = \text{step}(\sin(2.0\pi(t-750)*\text{tstep}/(\text{tstep}\cdot5160))) \]
\[ \text{fuelflow} = 0.00070 \text{ [kg/s]} \]
\[ \text{fuelinject3} = (\text{coscycle5} \cdot \text{inletstep}) \]
\[ \text{inletstep} = \text{step}(\sin(2.0\pi(t-1050)*\text{tstep}/(\text{tstep}\cdot5160))) - 0.5 \]
\[ \text{inletstep2} = \text{step}(\sin(2.0\pi(t+800)*\text{tstep}/(\text{tstep}\cdot5160))) - 0.5 \]
\[ \text{losssin} = \text{step}(\text{exp}((\text{step}(\text{coscycle2}) \cdot \text{coscycle2} - \text{step}(\text{coscycle2} - 0.50) \cdot (\text{coscycle2} - 0.50) \cdot 32) - 1) \cdot 1e4 \text{ [m}^{-1}] \]
\[ \text{losssinusoidal12} = (\text{coscycle4} \cdot \text{inletstep}) + 1 \cdot 1e4 \text{ [m}^{-1}] \]
\[ \text{lossthroat} = \text{step}(\text{exp}((\text{step}(\text{coscycle9}) \cdot \text{coscycle9} - \text{step}(\text{coscycle9} - 0.10) \cdot (\text{coscycle9} - 0.10) \cdot 32) - 1) \cdot 1e5 \text{ [m}^{-1}] \]
\[ \text{lossthroatsinusoidal1} = (\text{coscycle6} + 1) \cdot 1e4 \text{ [m}^{-1}] \]
\[ \text{lossthroatsinusoidal13} = (\text{coscycle6} + \text{coscycle10}) - 1 \cdot 1e4 \text{ [m}^{-1}] \]
\[ \text{mu0} = 1.846e-5\text{[kg m}^{-1}\text{ s}^{-1}] \]
\[ \text{temponx} = \text{exp}(0.512\pi(x/\text{x0}))*\text{step}(-x/\text{x0}+1000*(\text{step}(x/\text{x0})-\text{step}(x/\text{x0}-5)) - (300+700\text{exp}(-0.114*(x/\text{x0}-5)))\text{step}(x/\text{x0}-5)) \text{ [K]} \]
\[ \text{tlimit} = (900+3000\text{step}(1.1-D\text{sensor}/\text{Dset})) \text{ [K]} \]
\[ \text{tstep} = 1e-6 \text{[s]} \]
\[ \text{ttotal} = 5e-2 \text{[s]} \]
\[ \text{v3 = volumeAve(u)} @\text{throat valve} \]
\[ \text{vin = areaAve(u)} @\text{inlet area Side 2} \]
\[ \text{vout = areaAve(\text{Velocity u})@outlet area Side 1 1} \]
\[ \text{vref} = 1 \text{ [m s}^{-1}] \]
\[ \text{x0} = 0.0254 \text{ [m]} \]

END
Appendix C - Typical ANSYS Geometry, Mesh, CFX-Pre, Solver and Post Pages
# Appendix D – Reactant Mixing Results

## Reactant Mixing Results

<table>
<thead>
<tr>
<th>Reactant Mixing - 194 Hz - Single and Dual Fuel Injection - 0.5 to 3.0 grams/sec of Propane</th>
<th>Simulation</th>
<th>Frequency</th>
<th>Average Thrust</th>
<th>Average Fuel Flow</th>
<th>Isp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single Fuel Inject 0.5 grams/sec</td>
<td>194</td>
<td>10.476</td>
<td>0.0005</td>
<td>2127.107</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 1.0 grams/sec</td>
<td>194</td>
<td>31.331</td>
<td>0.001</td>
<td>3180.812</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 1.5 grams/sec</td>
<td>194</td>
<td>45.008</td>
<td>0.0015</td>
<td>3040.227</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 2.0 grams/sec</td>
<td>194</td>
<td>54.58675</td>
<td>0.002</td>
<td>2770.901</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 2.5 grams/sec</td>
<td>194</td>
<td>59.5095</td>
<td>0.0025</td>
<td>2416.629</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 3.0 grams/sec</td>
<td>194</td>
<td>61.7599</td>
<td>0.003</td>
<td>2090.014</td>
<td></td>
</tr>
<tr>
<td>Dual Fuel Inject 2.0 grams/sec (1.5 and 0.5 grams/sec)</td>
<td>194</td>
<td>51.0996</td>
<td>0.002</td>
<td>2593.888</td>
<td></td>
</tr>
<tr>
<td>Dual Fuel Inject 2.0 grams/sec (1 and 1.0 grams/sec)</td>
<td>194</td>
<td>52.865</td>
<td>0.002</td>
<td>2553.503</td>
<td></td>
</tr>
<tr>
<td>Dual Fuel Inject 1.5 grams/sec (1.125/0.375)</td>
<td>194</td>
<td>47.1614</td>
<td>0.0015</td>
<td>3191.973</td>
<td></td>
</tr>
<tr>
<td>Dual Fuel Inject 1.0 grams/sec (0.75/0.25)</td>
<td>194</td>
<td>17.848</td>
<td>0.002</td>
<td>905.9898</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 1.5 grams/sec (Fuel Vector Into Inlet Flow)</td>
<td>194</td>
<td>43.541</td>
<td>0.0015</td>
<td>2974.01</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 1.5 grams/sec (Fuel Vector Opposite Inlet)</td>
<td>194</td>
<td>43.541</td>
<td>0.0015</td>
<td>2974.01</td>
<td></td>
</tr>
<tr>
<td>Single Fuel Inject 1.5 grams/sec (Fuel Vector Perpendicular To Inlet Flow, Down)</td>
<td>194</td>
<td>47.8695</td>
<td>0.0015</td>
<td>3239.898</td>
<td></td>
</tr>
</tbody>
</table>
Appendix E – Sinusoidal Inlet and Throat Ball Valve Results

<table>
<thead>
<tr>
<th>Primary Fuel - Wedge (kg/sec)</th>
<th>Total Avg Engine Fuel (kg/sec)</th>
<th>Average Thrust (Newton)</th>
<th>lsp (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00006</td>
<td>0.00043</td>
<td>5.23</td>
<td>1240.67</td>
</tr>
<tr>
<td>0.0001</td>
<td>0.00071</td>
<td>17.44</td>
<td>2481.35</td>
</tr>
<tr>
<td>0.00012</td>
<td>0.00086</td>
<td>24.42</td>
<td>2894.91</td>
</tr>
<tr>
<td>0.00014</td>
<td>0.00100</td>
<td>30.70</td>
<td>3119.41</td>
</tr>
<tr>
<td>0.00018</td>
<td>0.00128</td>
<td>32.95</td>
<td>2603.88</td>
</tr>
<tr>
<td>0.00022</td>
<td>0.00157</td>
<td>36.59</td>
<td>2366.05</td>
</tr>
<tr>
<td>0.00024</td>
<td>0.00171</td>
<td>38.05</td>
<td>2255.61</td>
</tr>
<tr>
<td>0.00026</td>
<td>0.00186</td>
<td>40.59</td>
<td>2221.02</td>
</tr>
<tr>
<td>0.00028</td>
<td>0.00200</td>
<td>43.07</td>
<td>2188.41</td>
</tr>
<tr>
<td>0.00030</td>
<td>0.00214</td>
<td>44.13</td>
<td>2092.60</td>
</tr>
<tr>
<td>0.00032</td>
<td>0.00228</td>
<td>44.77</td>
<td>1990.25</td>
</tr>
<tr>
<td>0.00034</td>
<td>0.00243</td>
<td>42.21</td>
<td>1766.14</td>
</tr>
</tbody>
</table>
Appendix F – 2-D Nozzle Design Program

2-D Nozzle Design MATLAB Design Program

% Britton Jeffrey Olson
% Ph.D. Candidate
% Stanford University
% Department of Aero/Astro

% Introduction and Background
% This program gives the ideal nozzle geometry using the method of characteristics for a Quasi-2D Diverging Nozzle. Assume gas is exhausting from a combustion chamber that has no mass flow rate in. Using 2D nozzle flow relations, an optimal throat area is found that will produce the max amount of thrust for the given ambient pressure and combustion chamber parameters. This Area is automatically set and fed into the method of characteristics portion of that code. The method of characteristics also uses the exit Mach number that corresponds to the ideal exit area.

% Directions for running the program
% Run default or modify the problem parameters in nozzle.m and noz_cfd.m
% 1) Run the nozzle.m program- This will design the nozzle
% 2) Run noz_mesh.m- This will set up a grid for the newly designed nozzle
% 3) Run noz_cfd.m- This will solve the 2-D Euler equations on the nozzle mesh and plot the result.
% ***Note***
% You must run them in this order without clearing variables.
% Each script depends on the one before it. Also, solver has the functions used by noz_cfd.m and needs to be in the same directory.
% Figure(1): Static thrust as function of exit area
% Figure(2): Nozzle design and plots of Mach number & Pressure vs Length
% Figure(3): CFD simulation of designed nozzle

clear all;
clc;

% Problem parameters
T_c = 2000; % Temperature in the combustion chamber (K)
P_c = 1.2e6; % Pressure in the combustion chamber (Pa)
P_amb = 101e3; % Ambient pressure (Pa)
T_amb = 300; % Ambient temperature (K)
gamma = 1.25; % Ratio of Specific Heats Cp/Cv (Gamma)
W = 25.4; % Molecular weight of gas (kg/kmol)
width = .1; % Nozzle width (meters)
h_th = .025; % Throat height (meters)

% Method of Characteristics
num = 15; % Number of Characteristic lines
theta_i = .03; % Initial step in theta
plotter = 1; % Set to '1' to plot nozzle

dh = h_th/100;
max_iter = 10000;
R = 8314/W;

% Part A

%find where P becomes u
h(1) = h_th;
A_star = h_th*width;
M = 1;
dM1 = .1;
for i=1:max_iter
    h(i) = h(1) + (i-1)*dh;
    Ae(i) = h(i)*width;
    A_As = (Ae(i)/A_star)^2;
    A_ratio(i)=sqrt(A_As);
    %Newton Rhapson on Eq. 5.20 - Anderson text
    res = 1;
    if i > 1
        M = Ma(i-1);
    end
    while res > .001
        M2 = M + dM1;
        funa1 = -A_As + (1/M^2)*((2/(gamma+1))*(1+(gamma-1)*M^2/2))^(gamma+1)/(gamma-1);
        funa2 = -A_As + (1/M2^2)*((2/(gamma+1))*(1+(gamma-1)*M2^2/2))^(gamma+1)/(gamma-1);
        dv_dm = (funa2-funa1)/dM1;
        M = M - (funa1)/dv_dm;
        res = abs(funa1);
    end
    Ma(i) = M;
    % Find Pressure
    P(i) = P_c*(1+(gamma-1)*Ma(i)^2/2)^(-gamma/(gamma-1));
% Find thrust for each point
    Te(i) = T_c/(1+(gamma-1)*Ma(i)^2/2);
    Tt(i) = T_c/(1+(gamma-1)/2);
    Ve(i) = Ma(i)*sqrt(Te(i)*gamma*R);
    Vt(i) = sqrt(Tt(i)*gamma*R);
    rho(i) = P(i)/(R*Te(i));
    mdot(i) = rho(i)*Ve(i)*Ae(i);
    TT(i) = mdot(i)*Ve(i) + (P(i) - P_amb)*Ae(i);
    if P(i) < P_amb
        %break
        %Calculate the pressure if shock wave exists at the exit plane
        P_exit = P(i)*(1+(gamma*2/(gamma+1))*(Ma(i)^2-1));
        if P_exit <= P_amb
            P(i) = P_exit;
            break
        else
            end
        end
    end
figure(2)
plot(Ae,TT)
title('Thrust curve')
xlabel('Exit Area (m^2)')
ylabel('Thrust (N)')

% Part B
% Determine the nominal exit area of the nozzle
% to maximize thrust
[a,b]=max(TT);
% Over or Underexpand the nozzle
b = b;
A_max = Ae(b);
Max_thrust = TT(b);
hold on;
plot(A_max,Max_thrust,'r*')
legend('Thrust Curve','Max Thrust')

% Part C
% Method of Characteristics
M_e = Ma(b); %Mach number at ideal exit

% Find theta_max by using equation 11.33
theta_max = (180/pi)*(sqrt((gamma+1)/(gamma-1))*atan((sqrt((gamma-1)*(M_e^2-1)/(gamma+1))))-atan(sqrt(M_e^2-1)))/2;
% D_theta for each char line
del_theta = (theta_max - theta_i)/(num-1);
% Find
for i=1:num
  % Initialize mach number
  for j=1:num
    if i==1
      theta(i,j) = theta_i + del_theta*(j-1);
      nu(i,j) = theta(i,j);
      K_m(i,j) = theta(i,j) + nu(i,j);
      K_p(i,j) = theta(i,j) - nu(i,j);
    elseif i > 1
      K_p(i,j) = -K_m(1,i);
    % Find Thetas
    if j >= i
      theta(i,j) = del_theta*(j-1);
    else
      theta(i,j) = theta(j,i);
    end
    nu(i,j) = theta(i,j) - K_p(i,j);
    K_m(i,j) = theta(i,j) + nu(i,j);
  end
end
% Prandtl-Meyer function (using Newton Rhapson)
dM = .1; % Leave at about .1
if j == 1
  M_ex(i,j) = 1.00;
else
    M_ex(i,j) = M_ex(i,j-1);
end
M = M_ex(i,j);
res = 1;
while res > .01
    M2 = M + dM;
    funv1 = \(-nu(i,j)*(pi/180) + (sqrt((gamma+1)/(gamma-1))*atan((sqrt((gamma-1)*(M^2-1)/(gamma+1)))) - atan(sqrt(M^2-1))));
    funv2 = \(-nu(i,j)*(pi/180) + (sqrt((gamma+1)/(gamma-1))*atan((sqrt((gamma-1)*(M2^2-1)/(gamma+1)))) - atan(sqrt(M2^2-1))));
    dv_dm = (funv2-funv1)/dM;
    M = M - funv1/dv_dm;
    res = abs(funv1);
end
M_ex(i,j) = M;

% Find the angle mu
mu(i,j) = (180/pi)*asin(1/M_ex(i,j));

% Add last point to char line
theta(i,num+1) = theta(i,num);
nu(i,num+1) = nu(i,num);
K_m(i,num+1) = K_m(i,num);
K_p(i,num+1) = K_p(i,num);

char = zeros(num,num+1,2); for i=1:num
    for j=1:num+1
        % Draw points of intersection
        % Point 1 of all char lines
        if j == 1
            char(i,j,1) = 0;
            char(i,j,2) = h_th/2;
        end
        % Where first line hits the symmetry line
        if i == 1 & j==2
            char(i,j,1) = \(-h_th/2)/tan((pi/180)*(theta(i,j-1)-mu(i,j-1)));
            char(i,j,2) = 0;
        end
        % Where all other lines hit the symmetry line
        if j == i+1 & j>2
            char(i,j,1) = -char(i-1,j,2)/tan((pi/180)*(.5*theta(i,j-2)-.5*(mu(i,j-2)+mu(i,j-1))));
            char(i,j,2) = 0;
            test(i,j) = (theta(i,j-2)-.5*(mu(i,j-2)+mu(i,j-1)));
            testpty(i,j) = char(i-1,j,2);
            testptx(i,j) = char(i-1,j,1);
        end
        % All other data points for char 1 calculated
        if i == 1 & j>2 & j == i+1
            C_p = tan((pi/180)*.5*(theta(i,j-2)+theta(i,j-1))-.5*(mu(i,j-2)+mu(i,j-1)));
            C_m = tan((pi/180)*.5*(theta(i,j-1)+theta(i,j-1))-.5*(mu(i,j-1)+mu(i,j-1)));
            A = [1,-C_m;1,-C_p];
            B = [char(i,1,2) - char(i,1,1)*C_m; char(i,1,1)];
    end
end
char(1, j-1, 2) - char(1, j-1, 1)*C_p;
iterm(1, :) = inv(A)*B;
char(i, j, 1) = iterm(1, 2);
char(i, j, 2) = iterm(1, 1);
end

% All other points for all char lines calculated
if i > 1 & j~i+1 & j>2
C_p = tan((pi/180)*(.5*(theta(i, j-2)+theta(i, j-1))+.5*(mu(i, j-2)+mu(i, j-1))));
C_m = tan((pi/180)*(.5*(theta(i-1, j-1)+theta(i, j-1))-.5*(mu(i-1, j-1)+mu(i, j-1)))));
A = [1, -C_m, 1, -C_p];
B = [char(i-1, j, 2) - char(i-1, j, 1)*C_m; char(i, j-1, 2) - char(i, j-1, 1)*C_p];
iterm(1, :) = inv(A)*B;
char(i, j, 1) = iterm(1, 2);
char(i, j, 2) = iterm(1, 1);
end
end

% Fill in similar points (where char lines share points)
for i = 2 : num
for j = 2 : num
char(j, i, 1) = char(i-1, j+1, 1);
char(j, i, 2) = char(i-1, j+1, 2);
end
end

%******Make the nozzle shape and extend the char lines to wall******

% Initial start point of the nozzle (at throat)
noz(1, 1) = 0;
noz(1, 2) = h_th/2;

% Find all the points of the nozzle
for i = 2 : num
% Find different slopes and points to intersect
m1 = tan((pi/180)*(theta(i-1, num)+mu(i-1, num)));
if i == 2
m2 = (pi/180)*theta_max;
else
m2 = ((pi/180)*(theta(i-1, num+1)));
end
m3 = ((pi/180)*(theta(i-1, num)));
m4 = tan((m2+m3)/2);
A = [1, -m4; 1, -m1];
B = [noz(i-1, 2) - noz(i-1, 1)*m4; char(i-1, num+1, 2) - char(i-1, num+1, 1)*m1];
iterm(1, :) = inv(A)*B;
noz(i, 1) = iterm(1, 2);
noz(i, 2) = iterm(1, 1);

% Extend char lines to wall
char(i-1, num+2, 1) = noz(i, 1);
char(i-1, num+2, 2) = noz(i, 2);
end
%Last line
m1 = tan((pi/180)*(theta(num, num)+mu(num, num)));
m2 = ((pi/180)*(theta(num-1, num)));
m3 = ((pi/180)*(theta(num, num+1)));
m4 = tan((m2+m3)/2);
A = [1,-m4; 1,-m1];
B = [noz(num,2) - noz(num,1)*m4; char(num,num+1,2) - char(num,num+1,1)*m1];

iterm(1,:) = inv(A)*B;
noz(num+1,1) = iterm(1,2);
noz(num+1,2) = iterm(1,1);

%   Extend char lines to wall
char(num,num+2,1) = noz(num+1,1);
char(num,num+2,2) = noz(num+1,2);

if plotter == 1
%   Plot the nozzle shape
figure(1);clf;
subplot(2,1,1);
plot(noz(:,1),noz(:,2),'k','LineWidth',3)
hold on;
[a,b] = max(noz);
plot(a(1),A_max/width/2,'g*')
%
%   Plot for loop for char lines
for i = 1 : num
  figure(1)
  hold on;
  plot(char(i,:,1),char(i,:,2))
  hold on;
  plot(char(i,:,1),-char(i,:,2))
end
%
%   Plot the nozzle shape (bottom side)
figure(1)
subplot(2,1,1)
hold on;
plot(noz(:,1),-noz(:,2),'k','LineWidth',3)
hold on;
plot(a(1),-A_max/width/2,'g*')
title('Max Thrust (minimum length) Nozzle Design')
xlabel('Nozzle length (m)')
ylabel('Nozzle height (m)')
legend('Nozzle shape','Area_e_x_i_t(predicted)','Char. Lines')
else
end
%
%   Find % errors in A/A* and Mexit
error_Area = 100*(width*2*noz(num,2) - A_max)/(A_max)
error_Mach = 100*(M_e - M_ex(num,num))/M_e
%
%   Plot Mach Number and pressure through nozzle using the quasi-1D
% area relations.  (Isentropic expansion through nozzle)
Mnoz(1) = 1.0;  %   Choked Flow
for i=1: size(noz,1)
  A(i) = 2*noz(i,2)*width;
  A_Asq = (A(i)/A_star)^2;
  A_ratio(i) = sqrt(A_Asq);
  %Newton Rhapson on Eq. 5.20 = Anderson text
  res = 1;
  if i > 1
    M = Mnoz(i-1);
  while res > .001
M2 = M + dM1;
fun1 = A_Asq + (1/M^2)*((2/(gamma+1))*(1+(gamma-1)*M^2/2))^((gamma+1)/(gamma-1));
fun2 = A_Asq + (1/M2^2)*((2/(gamma+1))*(1+(gamma-1)*M2^2/2))^((gamma+1)/(gamma-1));
dv_dm = (fun2-fun1)/dM1;
M = M - fun1/dv_dm;
res = abs(fun1);
end
Mnoz(i) = M;

end
% Find Pressure
Pnoz(i) = P_c*(1+(gamma-1)*Mnoz(i)^2/2)^(-gamma/(gamma-1));
end

figure(1);
subplot(2,1,2)
plot(noz(:,1),Mnoz,'r*')
hold on;
plot(noz(:,1),Pnoz/P_amb,'b*')
hold on;
plot(noz(size(noz,1),1),M_e,'go')
hold on;
plot(noz(size(noz,1),1),1,'go')
xlabel('Nozzle length (m)')
ylabel('Mach number and P/P_a')
legend('Mach Number','P/P_a', 'M_e_x_i_t(predicted)','P_a_m_b/P_a_m_b')
## Appendix G – Thermodynamic Analysis Details

![Thermodynamic Analysis Table]

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>275 Maximum isp</td>
<td>2.28</td>
<td>106.8</td>
<td>4749.5</td>
<td>3.61</td>
<td>8.95</td>
<td>15.0</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>1355.0</td>
<td>700.0</td>
<td>800.0</td>
<td>6.3230</td>
</tr>
<tr>
<td>275 Maximum Thrust</td>
<td>12.13</td>
<td>191.6</td>
<td>1603.6</td>
<td>3.61</td>
<td>8.95</td>
<td>31.0</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>2540.0</td>
<td>1220.0</td>
<td>1490.0</td>
<td>6.3230</td>
</tr>
<tr>
<td>250 Maximum isp</td>
<td>2.28</td>
<td>96.7</td>
<td>4297.1</td>
<td>3.00</td>
<td>7.35</td>
<td>13.6</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>1680.0</td>
<td>910.0</td>
<td>1000.0</td>
<td>6.3760</td>
</tr>
<tr>
<td>250 Maximum Thrust</td>
<td>12.13</td>
<td>173.0</td>
<td>1447.5</td>
<td>3.00</td>
<td>7.35</td>
<td>39.2</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>2575.0</td>
<td>1210.0</td>
<td>1300.0</td>
<td>6.3760</td>
</tr>
<tr>
<td>225 Maximum isp</td>
<td>2.28</td>
<td>83.1</td>
<td>3694.0</td>
<td>2.57</td>
<td>6.48</td>
<td>13.2</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>1835.0</td>
<td>1045.0</td>
<td>1150.0</td>
<td>6.4200</td>
</tr>
<tr>
<td>225 Maximum Thrust</td>
<td>8.56</td>
<td>144.1</td>
<td>1708.8</td>
<td>2.57</td>
<td>6.48</td>
<td>24.2</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>400.0</td>
<td>2700.0</td>
<td>1550.0</td>
<td>1400.0</td>
<td>6.4200</td>
</tr>
<tr>
<td>200 Maximum isp</td>
<td>2.28</td>
<td>78.3</td>
<td>3392.5</td>
<td>2.30</td>
<td>5.75</td>
<td>12.7</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>375.0</td>
<td>1910.0</td>
<td>1070.0</td>
<td>1150.0</td>
<td>6.4520</td>
</tr>
<tr>
<td>200 Maximum Thrust</td>
<td>8.56</td>
<td>134.0</td>
<td>1588.2</td>
<td>2.30</td>
<td>5.75</td>
<td>22.2</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>375.0</td>
<td>2875.0</td>
<td>1540.0</td>
<td>1475.0</td>
<td>6.4520</td>
</tr>
<tr>
<td>100 Maximum isp</td>
<td>1.14</td>
<td>44.1</td>
<td>3920.2</td>
<td>1.50</td>
<td>3.75</td>
<td>6.9</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>370.0</td>
<td>1650.0</td>
<td>1065.0</td>
<td>1150.0</td>
<td>6.5750</td>
</tr>
<tr>
<td>100 Maximum Thrust</td>
<td>8.56</td>
<td>84.1</td>
<td>1005.2</td>
<td>1.50</td>
<td>3.75</td>
<td>14.0</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>370.0</td>
<td>2300.0</td>
<td>1275.0</td>
<td>1375.0</td>
<td>6.5750</td>
</tr>
<tr>
<td>Static Maximum isp</td>
<td>0.86</td>
<td>24.4</td>
<td>3216.6</td>
<td>1.0</td>
<td>2.5</td>
<td>5.1</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>370.0</td>
<td>1200.0</td>
<td>1180.0</td>
<td>1250.0</td>
<td>6.6910</td>
</tr>
<tr>
<td>Static Maximum Thrust</td>
<td>2.28</td>
<td>44.8</td>
<td>1990.3</td>
<td>1.0</td>
<td>2.5</td>
<td>9.1</td>
<td>1.1</td>
<td>1.1</td>
<td>300.0</td>
<td>370.0</td>
<td>2425.0</td>
<td>1495.0</td>
<td>1585.0</td>
<td>6.6910</td>
</tr>
<tr>
<td>Convective Speed (m/s)</td>
<td>Fuel Flow (g/s)</td>
<td>Avg Thrust (Newtons)</td>
<td>ISP (sec)</td>
<td>S2 (kJ/kg K)</td>
<td>S3 (kJ/kg K)</td>
<td>S4 (kJ/kg K)</td>
<td>S4' (kJ/kg K)</td>
<td>M1 (kJ/kg)</td>
<td>M2 (kJ/kg)</td>
<td>M3 (kJ/kg)</td>
<td>M4 (kJ/kg)</td>
<td>M4' (kJ/kg)</td>
<td>Eff Nozzle</td>
<td>Eff Comp</td>
</tr>
<tr>
<td>------------------------</td>
<td>-----------------</td>
<td>----------------------</td>
<td>----------</td>
<td>--------------</td>
<td>--------------</td>
<td>--------------</td>
<td>--------------</td>
<td>--------------</td>
<td>-------------</td>
<td>-------------</td>
<td>-------------</td>
<td>-------------</td>
<td>-------------</td>
<td>------------</td>
</tr>
<tr>
<td>275 Maximum ISP</td>
<td>2.28</td>
<td>106.8</td>
<td>4749.5</td>
<td>6.3530</td>
<td>7.5400</td>
<td>7.5400</td>
<td>7.6820</td>
<td>299.88</td>
<td>400.90</td>
<td>1464.34</td>
<td>713.97</td>
<td>822.92</td>
<td>0.85</td>
<td>0.7</td>
</tr>
<tr>
<td>275 Maximum Thrust</td>
<td>12.13</td>
<td>191.6</td>
<td>1603.6</td>
<td>6.3530</td>
<td>8.1660</td>
<td>8.1640</td>
<td>8.4040</td>
<td>299.88</td>
<td>400.90</td>
<td>3078.04</td>
<td>1303.72</td>
<td>1627.30</td>
<td>0.82</td>
<td>0.7</td>
</tr>
<tr>
<td>250 Maximum ISP</td>
<td>2.28</td>
<td>95.7</td>
<td>4297.1</td>
<td>6.4100</td>
<td>7.3300</td>
<td>7.1300</td>
<td>7.9330</td>
<td>299.88</td>
<td>400.90</td>
<td>1860.09</td>
<td>945.52</td>
<td>1047.79</td>
<td>0.89</td>
<td>0.7</td>
</tr>
<tr>
<td>250 Maximum Thrust</td>
<td>12.13</td>
<td>175.9</td>
<td>1447.5</td>
<td>6.4100</td>
<td>8.1510</td>
<td>8.2390</td>
<td>8.1540</td>
<td>299.88</td>
<td>400.90</td>
<td>2994.02</td>
<td>1291.92</td>
<td>1398.60</td>
<td>0.94</td>
<td>0.7</td>
</tr>
<tr>
<td>225 Maximum ISP</td>
<td>2.28</td>
<td>83.1</td>
<td>3694.0</td>
<td>6.4460</td>
<td>7.9810</td>
<td>7.9810</td>
<td>8.0950</td>
<td>299.88</td>
<td>400.90</td>
<td>2115.05</td>
<td>1100.49</td>
<td>1221.45</td>
<td>0.88</td>
<td>0.7</td>
</tr>
<tr>
<td>225 Maximum Thrust</td>
<td>8.56</td>
<td>144.1</td>
<td>1708.8</td>
<td>6.4460</td>
<td>8.2660</td>
<td>8.2660</td>
<td>8.3280</td>
<td>299.88</td>
<td>400.90</td>
<td>3155.75</td>
<td>1434.41</td>
<td>1518.41</td>
<td>0.95</td>
<td>0.7</td>
</tr>
<tr>
<td>200 Maximum ISP</td>
<td>2.28</td>
<td>76.3</td>
<td>3392.3</td>
<td>6.4150</td>
<td>8.0090</td>
<td>8.0090</td>
<td>8.0950</td>
<td>299.88</td>
<td>375.55</td>
<td>2146.38</td>
<td>1128.36</td>
<td>1221.45</td>
<td>0.91</td>
<td>0.7</td>
</tr>
<tr>
<td>200 Maximum Thrust</td>
<td>8.56</td>
<td>134.0</td>
<td>1588.2</td>
<td>6.4150</td>
<td>8.1790</td>
<td>8.2790</td>
<td>8.3910</td>
<td>299.88</td>
<td>375.55</td>
<td>3123.36</td>
<td>1446.37</td>
<td>1609.09</td>
<td>0.90</td>
<td>0.7</td>
</tr>
<tr>
<td>100 Maximum ISP</td>
<td>1.14</td>
<td>44.1</td>
<td>3920.2</td>
<td>6.5240</td>
<td>8.0020</td>
<td>8.0020</td>
<td>8.0950</td>
<td>299.88</td>
<td>370.49</td>
<td>1823.09</td>
<td>1122.58</td>
<td>1221.45</td>
<td>0.86</td>
<td>0.7</td>
</tr>
<tr>
<td>100 Maximum Thrust</td>
<td>8.56</td>
<td>84.3</td>
<td>1005.2</td>
<td>6.5240</td>
<td>8.2160</td>
<td>8.2160</td>
<td>8.3070</td>
<td>299.88</td>
<td>370.49</td>
<td>2640.57</td>
<td>1368.85</td>
<td>1448.34</td>
<td>0.94</td>
<td>0.7</td>
</tr>
<tr>
<td>Static Maximum ISP</td>
<td>0.86</td>
<td>24.4</td>
<td>3166.6</td>
<td>6.6400</td>
<td>8.1250</td>
<td>8.1250</td>
<td>8.1930</td>
<td>299.88</td>
<td>370.49</td>
<td>2008.90</td>
<td>1256.60</td>
<td>1339.20</td>
<td>0.89</td>
<td>0.7</td>
</tr>
<tr>
<td>Static Maximum Thrust</td>
<td>2.28</td>
<td>44.3</td>
<td>1990.3</td>
<td>6.6400</td>
<td>8.4080</td>
<td>8.4790</td>
<td>8.3280</td>
<td>299.88</td>
<td>370.49</td>
<td>2800.80</td>
<td>1633.40</td>
<td>1743.20</td>
<td>0.91</td>
<td>0.7</td>
</tr>
<tr>
<td>Convective Speed (m/s)</td>
<td>Fuel Flow (g/s)</td>
<td>Avg Thrust (Newton)</td>
<td>Isp (sec)</td>
<td>Hi</td>
<td>Mass Flow (g/s)</td>
<td>W Valves (m/s)</td>
<td>Eff Cycle w/ Periodic Compressor</td>
<td>Eff Cycle Constant Cp</td>
<td>Eff Cycle w/ Constantly Operating Compressor</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------------</td>
<td>----------------</td>
<td>---------------------</td>
<td>----------</td>
<td>----</td>
<td>----------------</td>
<td>----------------</td>
<td>-------------------------------</td>
<td>---------------------</td>
<td>--------------------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.75 Maximum isp</td>
<td>2.28</td>
<td>106.8</td>
<td>4749.5</td>
<td>256.6</td>
<td>41.0</td>
<td>2.44</td>
<td>0.47</td>
<td>0.58</td>
<td>0.36</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>275 Maximum Thrust</td>
<td>12.13</td>
<td>101.6</td>
<td>1603.6</td>
<td>254.6</td>
<td>220.0</td>
<td>0.45</td>
<td>0.49</td>
<td>0.50</td>
<td>0.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>250 Maximum isp</td>
<td>2.28</td>
<td>96.7</td>
<td>4297.1</td>
<td>256.6</td>
<td>41.0</td>
<td>2.44</td>
<td>0.46</td>
<td>0.52</td>
<td>0.38</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>250 Maximum Thrust</td>
<td>12.13</td>
<td>173.0</td>
<td>1447.5</td>
<td>256.6</td>
<td>220.0</td>
<td>0.45</td>
<td>0.56</td>
<td>0.58</td>
<td>0.51</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>225 Maximum isp</td>
<td>2.28</td>
<td>83.1</td>
<td>3694.0</td>
<td>256.6</td>
<td>41.0</td>
<td>2.44</td>
<td>0.44</td>
<td>0.50</td>
<td>0.37</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>225 Maximum Thrust</td>
<td>0.50</td>
<td>144.1</td>
<td>1708.8</td>
<td>256.6</td>
<td>155.0</td>
<td>0.65</td>
<td>0.54</td>
<td>0.55</td>
<td>0.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>300 Maximum isp</td>
<td>2.28</td>
<td>76.3</td>
<td>3102.5</td>
<td>267.4</td>
<td>41.0</td>
<td>2.44</td>
<td>0.46</td>
<td>0.50</td>
<td>0.41</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200 Maximum Thrust</td>
<td>0.56</td>
<td>134.0</td>
<td>1588.2</td>
<td>267.4</td>
<td>155.0</td>
<td>0.65</td>
<td>0.51</td>
<td>0.55</td>
<td>0.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100 Maximum isp</td>
<td>1.14</td>
<td>44.1</td>
<td>3920.2</td>
<td>269.6</td>
<td>21.0</td>
<td>4.76</td>
<td>0.34</td>
<td>0.40</td>
<td>0.29</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100 Maximum Thrust</td>
<td>0.56</td>
<td>84.8</td>
<td>1005.2</td>
<td>269.6</td>
<td>155.0</td>
<td>0.65</td>
<td>0.48</td>
<td>0.49</td>
<td>0.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Static Maximum isp</td>
<td>0.86</td>
<td>24.4</td>
<td>3116.6</td>
<td>269.6</td>
<td>21.0</td>
<td>4.76</td>
<td>0.34</td>
<td>0.38</td>
<td>0.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Static Maximum Thrust</td>
<td>2.28</td>
<td>44.8</td>
<td>1990.3</td>
<td>269.6</td>
<td>41.0</td>
<td>2.44</td>
<td>0.39</td>
<td>0.42</td>
<td>0.36</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Average Cycle Efficiency = 0.46

Average Constant Cp Efficiency = 0.51
Appendix H – Typical Valve Motor
<table>
<thead>
<tr>
<th>Type</th>
<th>GNM 2135</th>
<th>GNM 2145</th>
</tr>
</thead>
<tbody>
<tr>
<td>series</td>
<td>C</td>
<td>6000</td>
</tr>
<tr>
<td>nominal speed (rpm)</td>
<td>12</td>
<td>24</td>
</tr>
<tr>
<td>nominal voltage (V)</td>
<td>A</td>
<td>1.7</td>
</tr>
<tr>
<td>nominal current (A)</td>
<td>W</td>
<td>11</td>
</tr>
<tr>
<td>operation acc. to VDE 0530</td>
<td>free leads</td>
<td>reversible</td>
</tr>
<tr>
<td>insulation class acc. to VDE 0530</td>
<td>IP 21</td>
<td>IP 21</td>
</tr>
<tr>
<td>connection</td>
<td>reversible</td>
<td></td>
</tr>
</tbody>
</table>

**Mechanical Data**

| Mass moment of inertia (kg·m²) | 0.0052×10⁴ | 0.0043×10⁴ |
| Nominal torque (Nm) | 0.0175 | 0.0175 | 0.0255 | 0.0255 |
| Starting torque (Nm) | 0.16 | 0.21 | 0.21 | 0.21 |
| Max. continuous torque at stall (Nm) | 0.822 | 0.822 | 0.83 | 0.83 |
| Speed regulation constant (rpm) | 396 | 310 | 265 | 210 |
| Mechanical time constant (ms) | 13.2 | 10.5 | 12.1 | 9.4 |
| Field coil torque (Nm) | 0.888 | 0.888 | 0.888 | 0.888 |
| Motor weight (kg) | 0.879 | 0.879 | 0.879 | 0.879 |
| Motor weight (kg) | 0.375 | 0.375 | 0.355 | 0.355 |
| Bull bearing (6256/25) | 50 | 50 |
| F (allowable radial shaft load) | 20 | 20 |
|  |  |  |  |  |

**Electrical Data**

| Stator resistance (Ω) | 0.9 | 3.7 | 0.71 | 2.4 |
| Stator resistance (mΩ) | 0.63 | 2.7 | 0.4 | 1.45 |
| Terminal resistance (Ω) | 0.97 | 3.4 | 0.81 | 2.3 |
| Voltage constant (V/1000 rpm) | 0.6 | 3.4 | 1.8 | 3.4 |
| Torque constant (Nms/A) | 0.0103 | 0.0025 | 0.017 | 0.0025 |
| Starting current (A) | 12.4 | 7 | 14.8 | 16.4 |
| Max. peak current (A) | 16.3 | 8.8 | 23.8 | 12.5 |
| Electrical time constant (ms) | 0.065 | 0.78 | 0.49 | 0.68 |

**Thermal Data**

| MAX. ambient temperature (°C) | 40 | 40 | 40 | 40 |
| Insulation class acc. to VDE 0530 | F | F | F | F |
| Thermal time constant (min) | 15 | 15 | 15 | 15 |
| Temperature-rise without cooling (°C) | 5.8 | 5.6 | 6.4 | 6.4 |

Tolerances acc. to standard VDE 0530. ± 10% is valid for not VDE mentioned tolerances.

The values mentioned in the table are valid for supply with DC voltage with allowable harmonic content up to 2%. For undulatory current with increased harmonic content the rated motor values must be multiplied by 0.7.

The values are valid for operation in temperature-ranges from 0 to 40°C and it is not allowed to exceed them, even for a short-time, to avoid stator weakness.

**Motor design:**

Brushed 2-pole DC motor with permanent magnet field.

**Rotating direction:**

The rotating direction can be changed by inverting the connections.

1. **Order example**

   Motor:
   GNM 2130C
   24 V, 6000 rpm, 11 W

Special designs on request.
Appendix I – Typical Motor Batteries

<table>
<thead>
<tr>
<th>Packing</th>
<th>Powerizer 11.1V 1750mAh polymer Li-Ion Battery pack is made of 3 pcs of High Power Polymer Li-Ion (PL-683496K) cells. Wrapped by heavy duty heat shrink tube.</th>
</tr>
</thead>
</table>
| Voltage                     | 11.1 V (working) 12.6V (peak)  
  - Make your RC devices run faster due to 12.6V peak voltage supply! |
| Capacity                    | 1.75Ah (19.42Wh) Min, 1.8Ah (19.98Wh) Max                                                                    |
| Prewired                    | Charge terminal:  
  - 2.5" long 4 pin Female JST plug  
  - Discharge terminal:  
    - 4.0" long 14AWG wire open wire end  
      - Red = Positive  
      - Black = negative  
    - Included 1 pc plastic cap for prevent short circuit during shipment.  
    - You may buy connector for DIY |
| Max. Discharging Rate       | 43.75A (25.0C), Maximum continuous discharge, Recommended  
  - 87.5A (50.0C), Maximum Burst rate < 5 seconds |
| Dimensions (LxWxH)          | 106mm(4.2") x 36mm(1.4") x 22mm(0.9")                                                                     |
| Weight                      | 5.4Oz (150 grams)                                                                                           |
| Smart Tips                  | In order to keep the 11.1v 1750mAh polymer Li-ion pack balance while charging, please use either of the chargers below:  
  - Compact Battery Balance Charger for 7.4V-11.1V Li-Ion |


NiMH Battery Pack: Two 12V 1600mAh NiMH Battery (10x2/3A flat) Packs EDF

Your Price: $27.95
In Stock
Product ID #: 1637
Part Number: RA-H2/3A10I5WRx2

Quantity: 1
Buy
Add to a new shopping list
Email this page to a friend

Quantity Discounts

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Amount</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 to 20</td>
<td>$27.11</td>
</tr>
<tr>
<td>21 to 50</td>
<td>$26.55</td>
</tr>
<tr>
<td>51 to 100</td>
<td>$25.99</td>
</tr>
<tr>
<td>101 or more</td>
<td>$25.16</td>
</tr>
</tbody>
</table>

- 12 V, 1600 mAh battery pack made of 10 x2/3A high quality NiMH cells.
- Hi-power battery pack that can be discharged up to 10C (10Amp) rate.
- 5" 18 G Prewire (without connector) and ready to connects with any type of connector.
- Specially design for WATTAGE SABRE EDF RC PLANE. Also good for RC Robots
- Rapidly charge up, no memory effect and long cycle life.
- Dimension: 5-2/3" L x 1-3/8" W x 3/4" H
- Weight: 8 Oz
- Suggest use our 0.6A Compact smart charger to charge this battery. Charging time is 3.2 hours.
- Need 11.1V Advance Hi-Power Polymer Li-Ion pack for replace 12V NiMH RC pack? Please click here to order separately