ABSTRACT

ZHENG, FEI. Computational Investigation of High Speed Pulsjets. (Under the direction of Dr. William L. Roberts).

Pulsejet may be the simplest propulsion system ever. Due to its simplicity, the pulsejet may be an ideal low-cost micro-propulsion system. Although a considerable amount of work has been done on developing different types of pulsejets, understanding of the operation of these jets is still very limited. One goal of this investigation is to study the valveless pulsejet, find out effect of the geometry of inlet, combustion chamber and tailpipe on the performance of valveless pulsejet. Another goal is to study effect of the environment, such as free stream speed, ambient pressure and temperature, on the performance of pulsejet.

In this work, combined computational and experimental methods were used to study valved and valveless pulsejets to develop an understanding of how various inlets, exhaust sizes and coming flow speed effect the overall performance of the jet. Focus is put on the computational study. Fluid mechanics, acoustics and combustion were studied numerically to understand the operation of pulsejets and their interactive with the environment. The research objectives include the principles of valveless pulsejet design, optimize valveless pulsejet geometries and the effect of ejector and shroud.
Computational Investigation of High Speed Pulsejets

by

Fei Zheng

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

Mechanical Engineering

Raleigh, North Carolina
August 17, 2009

APPROVED BY:

Dr. William L. Roberts
Committee Chair

Dr. Andrey V. Kuznetsov
Committee Co-Chair

Dr. Terry D. Scharton

Dr. Tiegang Fang

Dr. Renzo Shamey
BIOGRAPHY

The author was born Fei Zheng of Suzhou, Jiangsu province, China in 1975, son of Dingchang Zheng and Yuanmei Li. He grew up in Taiyuan, Shanxi Province and graduated from The First High School of Taiyuan Railway in 1992.

Immediately following high school, Fei attended University of Sciences and Technology of China, where he received his Bachelor of Science in Thermal Science Engineering in July, 1997. After graduate, he got a job in Institute of Electrical Engineering, Chinese Academy of Sciences, where his research was focused on Solar Thermal energy for 8 years. During this period, he studied for his Master’s degree from 2000 and 2003, under the direction of Professor Anding Li, and received his Master degree in Electrical Engineering in July, 2003. In 2005 he came to North Carolina State University to pursue his Doctor degree in Mechanical Engineering under the direction of Dr. William Roberts and Dr. Andrey Kuznetsov.
ACKNOWLEDGMENTS

The ejector project was sponsored by the Ohio Aerospace Institute under the supervision of Dr. Daniel E. Paxson. Grant No. R-700-500048-40122

The author would like to thank his advisor Dr. William Roberts and co-advisor Dr. Andrey Kuznetsov for their help, guidance, patience, and encouragements during my study. The author would like to give his great appreciation to Dr. Terry Scharton his helpful comments and suggestions. The author would also like to thank Dr. Tiegang Fang and Dr. Renzo Shamey being my committee member.

The author would like to thank all the group members for their time and help.
# TABLE OF CONTENTS

## CHAPTER 1  Introduction

1.1 Pulsejet background ................................................................. 1  
1.1.1 Valved pulsejet ........................................................................ 1  
1.1.2 Valveless pulsejet .................................................................... 2  
1.1.3 Pulsejet cycles ......................................................................... 4  
1.2 Ejector ......................................................................................... 5  
1.3 Research objective ....................................................................... 7  
1.4 Combustion and fluid models used in simulations ................... 7  
  1.4.1 Turbulence model .................................................................... 8  
  1.4.2 Combustion model ................................................................. 10  

## CHAPTER 2  A New Acoustic Model for Valveless Pulsejets and Its Application to Optimization Thrust

2.1 Introduction .................................................................................. 14  
2.2 Experimental Setup ...................................................................... 16  
2.3 Numerical Model .......................................................................... 19  
2.4 Acoustic Model ............................................................................ 22  
2.5 Results and Discussion ................................................................ 24
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Section</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5</td>
<td>2.5.1</td>
<td>Static Environment</td>
</tr>
<tr>
<td>2.5</td>
<td>2.5.2</td>
<td>Pulsejet in Convective Stream</td>
</tr>
<tr>
<td>2.6</td>
<td></td>
<td>Conclusions</td>
</tr>
<tr>
<td>3</td>
<td>3.1</td>
<td>Introduction</td>
</tr>
<tr>
<td>3</td>
<td>3.2</td>
<td>Numerical model</td>
</tr>
<tr>
<td>3</td>
<td>3.3</td>
<td>Results and discussion</td>
</tr>
<tr>
<td>3</td>
<td>3.4</td>
<td>Conclusions</td>
</tr>
<tr>
<td>4</td>
<td>4.1</td>
<td>Introduction</td>
</tr>
<tr>
<td>4</td>
<td>4.2</td>
<td>Numerical model</td>
</tr>
<tr>
<td>4</td>
<td>4.3</td>
<td>Results and discussion</td>
</tr>
<tr>
<td>4</td>
<td>4.3.1</td>
<td>2 m Helmholtz pulsejet</td>
</tr>
<tr>
<td>4</td>
<td>4.3.2</td>
<td>50 cm Helmholtz pulsejet</td>
</tr>
<tr>
<td>4</td>
<td>4.4</td>
<td>Conclusions</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>Use Matlab simulation to fast evaluate Pulsejet performance</td>
</tr>
</tbody>
</table>

CHAPTER 3 Numerical study of the effect of inlet size and Interior obstructions on the performance of Valveless pulsejets

CHAPTER 4 Summary of the preliminary work on modeling the Helmholtz pulsejet engine

CHAPTER 5 Use Matlab simulation to fast evaluate Pulsejet performance
6.3.2 Multi-phase combustion and NO formation ........................................ 104

6.4 Conclusions .................................................................................................................. 106

CHAPTER 7 Numerical Study of a Pulsejet-Driven Ejector ............................................ 109

7.1 Introduction .................................................................................................................. 109

7.2 Numerical Model ......................................................................................................... 110

7.3 Results and Discussion ............................................................................................... 112

7.3.1 Diameter Optimization ........................................................................................ 116

7.3.2 Ejector length Effects ........................................................................................ 119

7.4 Conclusions ................................................................................................................ 121

APPENDIX .......................................................................................................................... 125

APPENDIX A: S-function in MATLAB for the 1-D pipe flow ........................................... 126
LIST OF FIGURES

Figure 1-1: Schematic of German Aeropulse used to power the V-1 ‘buzz bomb.’ ........... 1
Figure 1-2: Marconnet design of valveless pulsejet ............................................................ 3
Figure 1-3: Schubert design of valveless pulsejet ................................................................. 3
Figure 1-4: Escopette design of valveless pulsejet ............................................................... 3
Figure 1-5: Lockwood-Hiller design of valveless pulsejet .................................................... 4
Figure 2-1: Dimensions of the Experimental Valveless pulsejet (in cm) ......................... 17
Figure 2-2: Pulsejet cross-section showing details of fuel injection, pressure and temperature measurements .................................................................................................................. 17
Figure 2-3: Numerical model for the pulsejet ...................................................................... 20
Figure 2-4: Temperature distribution along the pulsejet wall ............................................. 21
Figure 2-5: Chamber pressure and temperature at port 3 for 1.6 cm inlet diameter and 50 cm exhaust duct length ................................................................................................................. 24
Figure 2-6: Frequency comparison between computational and experimental results ........ 25
Figure 2-7: Calculated inlet and exhaust frequency compared with pulsejet running frequency for 50 cm tail pipe. a) Computational data; b) Experimental data .................................................. 27
Figure 2-8: Inlet and exit temperatures vs. the inlet diameter for a 50 cm exhaust duct pulsejet .................................................................................................................................. 30
Figure 2-9: Temperature at the exit ...................................................................................... 30
Figure 2-10: Thrust vs. inlet diameter for 50 cm exhaust duct pulsejet .............................. 31
Figure 2-11: Thrust vs. inlet diameter .................................................................................. 32
Figure 2-12: Thrust vs. inlet length for 58 cm exhaust duct pulsejet ................................. 33
Figure 2-13: Chamber pressure 2.54 cm inlet diameter, 0.42 m exhaust duct pulsejet .......... 34
Figure 2-14: Thrust vs. inlet diameter for different free stream speeds ............................. 35
Figure 2-15: Thrust vs. inlet diameter. (a) 30 m/s free stream; (b) 120 m/s free stream ...... 36
Figure 3-1: Thrust and peak chamber pressure for different inlet diameter ....................... 47
Figure 3-2: Oxygen mass fraction in the pulsejet in an 80 m/s free stream environment with aerodynamic outer geometry .................................................................................................................. 49
Figure 3-3: Velocity vector in the pulsejet with obstruction placed at the throat ............... 50
Figure 3-4: Velocity vector in the pulsejet with obstruction placed at the inlet-chamber transition ........................................................................................................................................... 51
Figure 4-1: Helmholtz resonator .......................................................................................... 56
Figure 4-2: Computational domain for modeling the Helmholtz pulsejet engine ............ 58
Figure 4-3: generated for modeling the Helmholtz pulsejet engine .................................. 59
Figure 4-4: Temperature distribution along the pulsejet wall .......................................... 60
Figure 4-5: Air enters the combustion chamber of the 2 m Helmholtz pulsejet with 10.4 cm tailpipe diameter .................................................................................................................................................................................. 61
Figure 4-6: Pressure profile for the 2 m Helmholtz pulsejet with 10.4 cm tailpipe diameter 61
Figure 4-7: Chamber pressure for the 2 m Helmholtz pulsejet with different tailpipe diameters ........................................................................................................................................................................ 63
Figure 4-8: Temperature fields and velocity vectorfields for a 50 cm Helmholtz pulsejet with 9 cm tailpipe diameter ................................................................................................................................. 65
Figure 4-9: Chamber pressure of the 50 cm Helmholtz pulsejet ........................................ 66
Figure 4-10: Temperature field with two circular rings placed in the Helmholtz pulsejet combustion chamber to enhance turbulence and improve fuel and air mixing ...................... 67
Figure 4-11: Chamber pressure for the case with and without objects placed in the combustion chamber ............................................................................................................. 67
Figure 4-12: Combustion chamber pressure for different tailpipe length............................. 68
Figure 5-1: First layer of the Matlab simulation .................................................................... 77
Figure 5-2: The second layer of the Matlab simulation ......................................................... 78
Figure 5-3: The third layer, reaction in the combustion chamber ......................................... 80
Figure 5-4: Chamber pressure of the 50 cm Helmholtz pulsejet .......................................... 81
Figure 5-5: Chamber pressure of the 50 cm pulsejet with 10cm long and 1.6cm diameter inlet .............................................................................................................................................. 81
Figure 5-6: Simulation results for different free stream velocities ......................................... 83
Figure 5-7: ISP at different Altitude and Forward speed ....................................................... 85
Figure 5-8: effects of inlet geometry on the Total Thrust of valveless pulsejet ................. 86
Figure 5-9: Performance of valveless pulsejet with different inlet length and free stream velocities .................................................................................................................................. 87
Figure 6-1: Numerical model for the pulsejet ................................................................. 93
Figure 6-2: Chamber pressure for pulsejet in a 15 cm diameter shroud and open air ........ 98
Figure 6-3: Temperature of the flow field around the exit of the pulsejet....................... 98
Figure 6-4: Geometry of the ejector .................................................................................. 99
Figure 6-5: Temperature and pressure around the exit of the pulsejet with ejector. .......... 100
Figure 6-6: Pressure ratio vs. Bypass ratio ................................................................. 103
Figure 6-7. Steady state result for liquid JetA burning in pulsejet ................................. 104
Figure 6-8. Pulsejet running with liquid JetA of different diameters as fuel. ................. 105
Figure 6-9. Thrust and combustion chamber pressure change with liquid JetA diameter... 105
Figure 7-1. Mesh generation for combined pulsejet and ejector simulations .............. 111
Figure 7-2. Dimensions of the ejector ........................................................................ 113
Figure 7-3. Pressure distribution generated by the vortex ring ................................. 113
Figure 7-4. Moving average pressure and thruts for pulsejet with the base case ejector ... 115
Figure 7-5. Thrust on the ejector during one single cycle ........................................ 116
Figure 7-6. Vortex ring path and simulated cases ...................................................... 117
Figure 7-7. Thrust augmentation as a function of D/d .............................................. 118
Figure 7-8. Thrust augmentation as a function of x/d .............................................. 119
Figure 7-9. Pressure oscillations in the ejector ......................................................... 120
Figure 7-10. Thrust augmentation as a function of L/d ........................................... 121
LIST OF TABLES

Table 2-1: Chamber temperature from simulations and experimental data.......................... 26
Table 2-2: Frequencies obtained from experimental data versus those obtained from the analytical model ................................................................................................................ 28
Table 2-3: Frequencies obtained from a computer simulation versus those obtained from the analytical model ................................................................................................................ 29
Table 2-4: Simulation results for smallest and largest diameters for 42 cm exhaust duct pulsejet ................................................................................................................................. 34
Table 2-5: a get from 50 and 80 m/s results ........................................................................ 36
Table 3-1: Total force of pulsejet with different inlet geometry in 30 m/s free stream ......... 48
Table 3-2: Total force of pulsejet with different inlet geometry in 50 m/s free stream ........ 48
Table 4-1: Average inhale velocity for 2 m Helmholtz pulsejet with different tailpipe diameters ................................................................................................................................. 62
Table 4-2: performance of 50 cm pulsejets in different types ............................................. 69
Table 6-1: Results for a pulsejet running in shroud with deferent geometries ................. 101
Table 6-2: Results for pulsejet running in 15 cm diameter, with different boundary conditions ................................................................................................................................. 102
CHAPTER 1. INTRODUCTION

1.1 Pulsejet background

1.1.1 Valved pulsejet

The pulsejet is an unsteady propulsion device that generates intermittent thrust [1]. The concept of a pulsejet can be traced back to the beginning of the 20\textsuperscript{th} century. Two French engineers, Esnault and Peltrie, patented a design of an engine that drove a turbine wheel. However, the first working pulsejet was designed by German engineer Paul Schmidt, which was called the Schmidt tube, in the 1930’s. This jet was used to power the German V-1 weapon, “Buzz Bomb”, in the World War II. Figure 1-1 shows the sketch of V-1 “Buzz
Bomb”, which was the first practical application of the pulsejet. It can be viewed as a prototype of a modern cruise missile.

This pulsejet used a series of reed valves at the intake end of the tube to intake a volume of fresh air to mix with the atomized fuel prior to ignition, thus, it is called valved pulsejet. In valved pulsejets, reed valves allow the injection of fresh reactants but prevent hot gases from leaving through the inlet. However, the reed valves operating in high temperature and high frequency are subjected to intense physical and thermal punishment. They are good for only 20-30 minutes continuous operation, thus regular replacement is required. It is good for a one-time use missile but when it comes to a longer time application, the only moving part of the valved pulsejet, the reed valves, comes to be a weak part of the system.

1.1.2 Valveless pulsejet

By proper utilization of wave processes in an inlet duct of adequate length, the amount of negative thrust can be minimized in a valveless pulsejet [2].

In 1909, Marconnet developed what he termed an aerodynamic valve that replaced the reed valves of conventional design (Figure 1-2). The objective of the aerodynamic valve was to allow the intake of air for oxidation but, at the same time, offer resistance to backflow momentum created by the combustion event. This design is the grandfather of all valveless pulsejets.

The principle of the valveless pulsating combustor was rediscovered by Lt. William Schubert of the US Navy in the early 1940s. His design, called the “resojet” at the time on account of
its dependence on resonance, is one of the simplest successful valveless designs of all (Figure 1-3) [3].

Figure 1-2: Marconnet design of valveless pulsejet

Figure 1-3: Schubert design of valveless pulsejet

There were a few other designs of valveless pulsejets after World War II. The Escopette, illustrated in Figure 1-4, was a valveless designed in French France in the early 1950's. To avoid negative thrust from inlet, inlet was made rearward in this designation.

Figure 1-4: Escopette design of valveless pulsejet

The United States government made considerable advances in valveless pulsejet technology in the 1950's to early 60's. Project Squid [4-8], a collaboration between the United States Navy and Air Force to research and develop all potential sources of jet propulsion available at that time, became greatly involved in the investigation of valveless pulsejets. It has been suggested that the Logan jet improved the rate of heat release and cycle efficiency.
However, such a design differed from pulsejets of the present work in that air was directly injected during operation rather than accomplishing air intake through self-aspiration.

Another project supported by US government focusing on the investigation of valveless pulsejet reactors was also conducted between 1961 and 1963. Numerous valveless designs were developed, produced, and tested in the course of the project. The most well known one of these designs was the Lockwood-Hiller pulsejet shown in Figure 1-5.

![Figure 1-5: Lockwood-Hiller design of valveless pulsejet](image)

1.1.3 Pulsejet cycles

The ideal pulsejet is based on the Humphrey thermodynamic cycle, where isochoric heat addition (combustion) follows an isentropic compression and isobaric heat rejection follows an isentropic expansion. The reactants enter the combustion chamber when the pressure in the combustion chamber is lower than the ambient pressure. Residual hot gases and heat transfer from the hot walls raise the reactants’ temperature to the auto-ignition temperature, initiating combustion of the reactants. The ensuing heat release increases pressure, and the combustion gases then expand down the exhaust duct and exit at high velocity, generating
thrust. Once the combustion gases at the exit have expanded to nearly atmospheric pressure, the overexpansion due to momentum of the combustion products causes the pressure to decrease in the combustion chamber to sub-atmospheric; also, the reflected wave propagates back up the exhaust duct towards the combustion chamber. When the reflected wave reaches the chamber, the combustion chamber pressure increases again, auto ignition occurs, and the cycle repeats itself.

In reality, the combustion process is not isochoric, and this is especially true for the valveless pulsejet. Because there is no mechanical compression, the peak pressure is low and, therefore, the overall thermodynamic efficiency is low. Because of their relatively low efficiency, pulsejets were surpassed by the much more efficient turbojets and were put aside for decades following limited development into the early 1960’s. In a turbojet, high efficiency is achieved through high compressor pressure ratios. However, the compressor is a complex and expensive piece of turbo machinery that consumes a large fraction of the chemical enthalpy. As the scale of turbo machinery is reduced, the compressor efficiency decreases and, therefore, the efficiency of a turbojet drops nonlinearly with size when it is utilized in a micro-propulsion system. Therefore, there may be applications where small scale, simplicity, low cost, and high thrust-to-weight ratio make the pulsejet attractive.

1.2 Ejector

Ejector or thrust augmentor is separate device which is placed in the exhaust of a jet engine to increase the amount of thrust produced. For low speed applications, high speed, high
temperature exhaust is not optimal, would prefer lower temperature and speed with larger mass flux to provide higher thrust.

Kinetic Energy of the flow:

\[
\dot{E} = \frac{1}{2} \dot{m} v^2
\]  \hspace{1cm} (1-1)

Momentum of the flow:

\[
F = \dot{m} v = \sqrt{2 \dot{m} \dot{E}}
\]  \hspace{1cm} (1-2)

From equation (1-2), could find that for the same kinetic energy, higher mass flow provide higher thrust. In static or low speed environment, large mass flow and lower velocity is desired.

There are numerous designs of ejectors, the simplest form of an ejector is just a pipe have a larger diameter than the jet tailpipe. The function of the ejector is to transfer energy from high temperature, high speed primary flow from jet to low temperature, low speed secondary flow. For a steady flow ejector, this energy transfer process is via viscous and turbulence shear stress. It is not efficient, due to the irreversible nature of the process.

For unsteady flow ejector, however, pressure interface was generated and thus push the secondary flow through the ejector, which makes the unsteady flow ejector quite efficient in thrust augment. Pulsejet is a typical source of the unsteady flow. While the flow from the pulsejet gets through the ejector, it is mixed with the ambient air, velocity decreased and
mass flow rate increased. A proper designed ejector could maximize the mass flow rate through the ejector, thus increase the thrust generated.

For pulsed flow from the pulsejet, another theory is that vertex generated at the exit of the pulsejet created low pressure region. This low pressure region moves to the ejector together with the vortex. When the low pressure region hit the ejector, thrust was generated [9].

1.3 Research objective

The pulsejet is driven by acoustic, which makes it very sensitive to the geometries of the combustion chamber, tailpipe and inlet. Although the engine is simple, it is not practice to generate pulsejet with all dimensions to decide the optimal one, not to say the difficulty to test the pulsejet in different environments. Experimental also have difficulty to collect detailed results of the flow field inside the pulsejet.

This work is to generate a valid CFD model for pulsejet simulation and use the model to predict the pulsejet performance. A detailed analysis will be carried out to find out how the geometry and environment affect the performance of the pulsejet. The object is to develop the principle to design a pulsejet and find out optimal pulsejet for different purpose and under different environments.

1.4 Combustion and fluid models used in simulations

In this work, most of the simulations were made by commercial available CFD software CFX. Models were chosen from the models provided by the software. However, detailed
knowledge of the principle and performance of the models is necessary to understand the simulation results and their limits and errors.

1.4.1 Turbulence model

To model the flow in the pulsejet simulation, \( k-\varepsilon \) 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 [10].

\( k \) 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. \( m^2/s^2 \). \( \varepsilon \) 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. The continuity equation is then:

\[
\frac{d\rho}{dt} + \nabla \cdot (\rho U) = 0
\]

and the momentum equation becomes:

\[
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) - \nabla \cdot (\mu_{\text{eff}} \nabla U) = \nabla p' + \nabla \cdot (\mu_{\text{eff}} \nabla U)' + B
\]
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

\[
\mu_{\text{eff}} = \mu + \mu_t \quad (1-5)
\]

\[
p' = p + \frac{2}{3} \rho k 
\quad (1-6)
\]

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 via the relation

\[
\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon} \quad (1-7)
\]

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:

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho U k) = \nabla \cdot \left( \mu + \frac{\mu}{\sigma_k} \right) \nabla k + P_k - \rho \varepsilon \quad (1-8)
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left( \mu + \frac{\mu}{\sigma_\varepsilon} \right) \nabla \varepsilon + \frac{\varepsilon}{k} \left( C_{\varepsilon_1} P_k - C_{\varepsilon_2} \rho \varepsilon \right) \quad (1-9)
\]

where \( C_{\varepsilon_1}, C_{\varepsilon_2}, \sigma_{k} \) and \( \sigma_{\varepsilon} \) are constants. \( P_k \) is the turbulence production due to viscous and buoyancy forces, which is modeled using:
\[ P_k = \mu_r \nabla U \cdot \left( \nabla U + \nabla U^T \right) - \frac{2}{3} \nabla \cdot U (3 \mu_r \nabla \cdot U + \rho k) + P_{lb} \] (1-10)

1.4.2 Combustion model

In this work, most simulations used propane as the fuel. The gas phase fuel made the simulations need not deal with the multi-phase flow problem, which made the simulation quicker and easier to get more accurate results. Propane is also easy to be handled in experiments, whose results are needed to validate the simulation results.

The model used propane-air 5-step reaction model, provided by CFX package, to simulate the combustion of propane. The 5 reactions included are propane oxidation, Carbon monoxide oxidation, Hydrogen oxidation, forward and backward water-gas reactions.

In this work, the chemical reaction rate is fast relative to the transport processes in the flow, Eddy Dissipation model is thus used to simulate the combustion process.

The eddy dissipation model is based on the concept that chemical reaction is fast relative to the transport processes in the flow. When reactants mix at the molecular level, they instantaneously form products. The model assumes that the reaction rate may be related directly to the time required to mix reactants at the molecular level. In turbulent flows, this mixing time is dominated by the eddy properties, and therefore, the rate is proportional to a mixing time defined by the turbulent kinetic energy, \( k \), and dissipation, \( \varepsilon \).

\[ \text{Rate} \propto \frac{\varepsilon}{k} \] (1-11)
This 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 [11].
REFERENCES


CHAPTER 2. ACOUSTIC MODEL FOR VALVELESS PULSEJETS AND ITS APPLICATION TO OPTIMIZE THRUST

Due to its simplicity, the valveless pulsejet may be an ideal low cost propulsion system. In this paper, a new acoustic model is described, which can accurately predict the operating frequency of a valveless pulsejet. Experimental and computational methods were used to investigate how the inlet and exhaust area and the freestream velocity affect the overall performance of a 50 cm pulsejet. Pressure and temperature were measured at several axial locations for different fuel flow rates and different geometries. Computer simulations were performed for exactly the same geometries and fuel flow rates using a commercial CFD package (CFX) to develop further understanding of the factors that affect the performance of a valveless pulsejet. An acoustic model was developed to predict the frequency of these valveless pulsejets. The new model treats the valveless pulsejet engine as a combination of a Helmholtz resonator and a wave tube. This new model was shown to accurately predict geometries for maximum thrust. The model was further extended to account for the effect of freestream velocity. Evidence is provided that valveless pulsejet generates the highest thrust when the inherent inlet frequency matches the inherent exhaust frequency.
2.1 Introduction

The pulsejet is an unsteady propulsion device that generates intermittent thrust [1]. The reactants enter the combustion chamber when the pressure in the combustion chamber is lower than the ambient pressure. Residual hot gases and heat transfer from the hot walls raise the reactants’ temperature to the autoignition temperature, initiating combustion of the reactants. The ensuing heat release increases the pressure, and the combustion gases then expand down the exhaust duct and exit at high velocity, generating thrust. Once the combustion gases at the exit have expanded to nearly atmospheric pressure, overexpansion due to the momentum of the combustion products causes the combustion chamber pressure to decrease to subatmospheric, causing fresh reactants to be pulled into the combustion chamber, and the process to begin again.

Simultaneously, there is an acoustic wave propagating down the exhaust duct, reflecting off the exhaust exit, traveling back up to exhaust duct as an expansion wave. The expansion wave reflects off the inlet and travels back down the exhaust, reflecting off the exhaust exit (now as a compression wave) and propagating back to the combustion chamber. When the reflected wave reaches the chamber, the combustion chamber pressure increases again, the autoignition occurs, and the cycle repeats itself.

The primary reason for the development of a valveless pulsejet is that in most designs, the use of reed valves limits the reliability and longevity of the engine, and renders the pulsejet difficult to scale down in size. In the valved pulsejet, the function of the reed valves is to
prevent a reversal of the flow at the inlet—and therefore negative momentum transport—when the combustion chamber pressure exceeds the freestream stagnation pressure. By proper utilization of wave processes in an inlet duct of adequate length, the amount of negative thrust can be minimized in a valveless pulsejet [2].

In 1909, Georges Marconnet developed the first pulsating combustor without valves (Fig. 1-2), and it is the grandfather of all valveless pulsejets. The US government made considerable advances in valveless pulsejet technology in the 1950s and early 1960s. Project Squid [3–7], a collaboration between the United States Navy and Air Force to research and develop all potential sources of jet propulsion available at that time, became greatly involved in the investigation of valveless pulsejets.

Another project supported by the US government focusing on the investigation of valveless pulsejet reactors was also conducted between 1961 and 1963 [8]. Numerous valveless designs were developed, produced, and tested in the course of the project. The most well known of these designs was the Lockwood–Hiller pulsejet shown in Fig. 1-5. It uses the same principles of the Marconnet design; however, the entire tube was bent into a U-shape to have thrust going in the same direction from both the inlet and exhaust.

Due to their relatively low efficiency, pulsejets were surpassed by the much more efficient turbojets and were put aside for decades following their development into the early 1960s. As the scale of turbo machinery is reduced, the compressor efficiency decreases and, therefore, the efficiency of a turbojet nonlinearly drops with its size when it is utilized in a
micropropulsion system [9]. Therefore, there may be applications where small scale, simplicity, low cost, and high thrust-to-weight ratio make the pulsejet attractive.

Recently, pulsejets have been studied utilizing modern numerical techniques [10–12]. Geng et al. [12] performed a combined experimental and computational study on a small scale valveless pulsejet. In Ref. [12], the area ratio of the inlet to that of the combustion chamber and the length ratio of the inlet and exhaust pipes were used to evaluate the scalability of the valveless pulsejet. However, the physical reasons why the inlet area is related to the length ratio of inlet and exhaust pipe length were not investigated.

The purpose of this paper is to develop a model to predict the operating frequency of a valveless pulsejet and suggest how inlet geometry affects the performance of a valveless pulsejet. This is accomplished by comparing the performance of a valveless pulsejet with different geometries to determine how inlet and exhaust geometry affects frequency, temperature, and thrust of the valveless pulsejet, leading to an optimization of the design and a set of design rules.

2.2 Experimental Setup

The pulsejet discussed in this paper is a modified version of the Bailey machining service (BMS) hobby-scale (50 cm in length) pulsejet where the original valved inlet is replaced with a straight pipe (Figure 2-1).
As shown in Figure 2-1, the pulsejet is composed of three parts: the valveless head, body, and extension. Ports 1–5 were added to allow temperature and pressure measurements (Figure 2-1).

![Figure 2-1: Dimensions of the Experimental Valveless pulsejet (in cm)](image)

<table>
<thead>
<tr>
<th>Port</th>
<th>Dimensions (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>9.6</td>
</tr>
<tr>
<td>1</td>
<td>7.6</td>
</tr>
<tr>
<td>3</td>
<td>6.8</td>
</tr>
<tr>
<td>4</td>
<td>44</td>
</tr>
<tr>
<td>5</td>
<td>29.6</td>
</tr>
</tbody>
</table>

**HEADS:**
- Diameter = 1.3, 1.6, 2.2, 2.5

**BODY:**
- Diameter = 6.2 & 3.2

**EXTENSIONS:**
- Diameter = 3.2;
- Length = 7.6, 15.2, 22.9

![Figure 2-2: Pulsejet cross-section showing details of fuel injection, pressure and temperature measurements](image)

Figure 2-2 shows the structure of the temperature and pressure sensor on the pulsejet and how the fuel was injected into the combustion chamber. The pulsejet was secured to a low-friction linear bearing assembly. Four valveless heads were made with different diameters (1.3 cm, 1.6 cm, 2.2 cm, and 2.5 cm) and different lengths (2.54 cm, 5.08 cm, and 7.62 cm)
to determine how inlet geometry affects performance. Fuel was continuously fed into the combustion chamber via a 3 mm stainless steel tube with eight small holes drilled into it and located at the same axial position as Port 1, but orthogonal to it, as shown in Figure 2-2. Various extensions were added to the exhaust to evaluate the contribution of exhaust duct length to the valveless pulsejet.

Propane was used as the fuel to run the pulsejet for all tests. This was done to simplify the comparison between the test results and the simulations by eliminating the need for multiphase modeling. Fuel was directly fed into the combustion chamber at a constant flow rate of 0.47 g/s during all experiments. Three Type B (platinum and platinum/rhodium) thermocouples were used to measure the time-averaged temperature simultaneously at Ports 1, 3, and 5 along the jet axis. These thermocouples (TCs) were custom made, and run through ceramic material, thus placing the TC junction at the centerline of the pulsejet. The voltage was measured with a Pentium 4 computer via a data acquisition card with a maximum input of 4 Hz per channel. This measured temperature is a time-averaged value due to the high operation frequency of the pulsejet and thermal inertia of the thermal couples. Also, (time-averaged) inlet and exhaust temperatures were measured at the inlet plane and exhaust exit plane, respectively. Both time-averaged and instantaneous pressures were measured at Ports 1, 3, and 5 on the jets. To measure the time-averaged pressure in the combustion chamber, a mercury manometer was used. For instantaneous pressure, a Kulite XTE-190-5G pressure transducer was used. This transducer has a peak pressure reading ability of 350 kPa with 0.1
kPa resolution and 300 kHz sampling frequency. The operating frequency of the pulsejet was determined from the oscillation frequency of the instantaneous pressure [13].

2.3 Numerical Model

Numerical simulations of acoustic resonators are addressed in numerous publications [10–12, 14, 15]. In this paper, the commercially available CFD software, CFX™ 5.7 package, was used to model the combustion and flow inside and outside of the pulsejet. Because the pulsejet is symmetric about the axis, the geometry was simplified to two-dimensional axisymmetric to save computational time. The mesh was generated in a 4 deg slice instead of the full 360 deg region. In Figure 2-3, the computational domain is different from Ref. [13], which only simulated the flow field inside the pulsejet. In this paper, as shown in Figure 2-3, the computational domain includes not only the interior of the pulsejet but also an extended domain to compute the flow field around the pulsejet. Because the boundaries were set far from the pulsejet, effects from pulsejet operation were negligible at the boundaries, and the boundary condition was set to temperature $T_0$ equals to 300 K and pressure $P_0$ equals to $10^5$ Pa.

A much smaller node distance was used for the flow field inside the pulsejet, shown in Figure 2-3, than for the flow field outside the pulsejet, with the total number of nodes in this model being about 18,000. The code was also tested with a double mesh density case. However, results showed that further increasing the density of nodes does not affect the results. The computations were performed on the NCSU IBM Blade Center utilizing a single
3.0 GHz Inter Xeon processor. Typical computational time for one cycle of the pulsejet was about 18 CPU hours.

A $k-\varepsilon$ model based on the Reynolds averaged Navier–Stokes (RANS) equations was used because it offers a good compromise in terms of accuracy and robustness [16, 17]. The eddy dissipation model was chosen as the combustion model. A propane-air five step reaction
mechanism (propane oxidation, hydrogen oxidation, CO oxidation, and both directions of water-gas shift reaction) provided by CFX 5.7 was used to simulate the combustion process.

There is a significant heat flux between the pulsejet walls and fluid inside the pulsejet. However, the frequency is high enough and the thermal inertia is large enough so that the wall temperature distribution can be assumed to be steady. To find the steady temperature distribution, a steady state simulation was performed, where boundary conditions were based on the average gas temperature in a transit simulation. This temperature distribution was further simplified to what is shown in Figure 2-4, where the region between two dashed lines represents the combustion chamber, with a constant temperature of 1000 K along the combustion chamber and then an exponential decay toward both the inlet and exit planes, which are at 400 K.

![Figure 2-4: Temperature distribution along the pulsejet wall](image)

The computational procedure for the pulsejet was initiated by setting the velocity to 0 m/s and temperature to 300 K everywhere in the computational domain. A stoichiometric mixture of air and propane was then injected into the combustion chamber and ignited. This generated a high pressure, and pulsed operation of the jet was initiated. This process is quite similar to pulsejet start-up in the analogous experiments.
It was also of interest to understand how the freestream velocities affected the operation of valveless pulsejets. Pulsejets flying at different forward flight speeds were modeled by changing the boundary condition at the inlet of the enclosure flow field. The freestream velocity was varied between 0 m/s zero and 80 m/s.

Monitors for pressure and temperature were set at precisely the same locations as in experiments along the centerline of the pulsejet. Data were acquired at every time step (10^{-5} s) during the simulation. Time-averaged temperature values were calculated by averaging the temperature over a cycle for comparison with experimental (TC) data.

2.4 Acoustic Model

Self-compression is one of the most important characteristics of pulsejets, so understanding this phenomenon is of considerable interest. Pressure oscillations in a pulsejet are amplified by an acoustic resonance. Traditionally, a valved pulsejet is modeled as a tube closed at the valve end and open at the exhaust end, so the fundamental resonance occurs when the total length of the pulsejet is equal to one-quarter of an acoustic wavelength. However, an acoustic investigation of valved pulsejets revealed that, because of the increased diameter of the combustion chamber, the frequency of a valved pulsejet obeys the following equation [18, 19]:

\[
\frac{\omega_j V}{S_c c_e} \tan \frac{\omega_j L_e}{c_e} = 1
\]  

(2-1)
where $\omega_e = 2\pi f_e$, $f_e$ is the resonate frequency associated with the exhaust pipe, $c_e$ is the average speed of sound in the exhaust tube (spatial and temporal average), $S_e$ is the cross section area of the exhaust pipe, $V$ is the volume of the combustion chamber, and $L_e$ is the length of the exhaust pipe.

The inlet length was much shorter than the exhaust, only a few diameters long, and thus much shorter than the sound wavelength. The volume of the inlet is also small compared to the volume of the combustion chamber. Therefore, the combination of this short inlet and combustion chamber is modeled as a Helmholtz resonator. The frequency of a Helmholtz resonator is calculated as [19]

$$f_i = \left[\frac{c_i}{\pi} \left(\frac{S_i}{V(L_i)}\right)\right]^{0.5} \tag{2-2}$$

where $f_i$ is the resonate frequency associated with the inlet, $c_i$ is the average speed of sound in the inlet and combustion chamber, $S_i$ is the cross section area of the inlet, $V$ is the volume of the combustion chamber, and $L_i$ is the length of the inlet. Since the reactant mixture in the pulsejet has similar heat ratio and specific gas constant to that of air, in this paper, the speed of sound was calculated by assuming a $\gamma$ (at the appropriate average temperature) and $R$ for air.

Since the operating frequency of the pulsejet is affected by both the inlet and exit tubes, it is expected to be a function of a compound variable that combined both the inlet and exhaust frequencies. In this research, the operating frequency of a pulsejet $f$ was postulated to be the average value of the inlet and exhaust frequency:

$$f = \frac{f_i + f_e}{2}$$
\[ f = (f_i + f_e)/2 \]  

(2-3)

where \( f_i \) and \( f_e \) are inlet and exhaust frequencies defined in Eqs. (2-1) and (2-2), respectively.

2.5 Results and Discussion

2.5.1 Static Environment

The instantaneous pressure at Port 3 is shown in Figure 2-5 for a pulsejet with a 1.6 cm inlet diameter and 50 cm exhaust duct length. From this comparison between experimental and computational results, two effects are observed. First, the amplitude of pressure variation is almost the same in experimental and computational data. Second, the frequency observed in computational data is a little larger than that in experimental data. Figure 2-5 also shows computationally obtained temperature at Port 3; the time-averaged temperature at Port 3 is 2000 K.

![Figure 2-5: Chamber pressure and temperature at port 3 for 1.6 cm inlet diameter and 50 cm exhaust duct length](image-url)
Operational frequencies for both experimental and computational are relatively accurately obtained by taking the average value for several cycles. This frequency is generated by both the acoustic waves traveling in the tube and the gases flowing into and out of the tube due to the large pressure differences. Therefore, the pulsejet frequency contains information about all the properties relative to combustion, flow, and heat transfer. As shown in Figure 2-6, both experimental and computational results show a frequency increase with an increase of inlet diameter and decrease with an increase of the exhaust duct length. Both of these trends are consistent with our modeling of the inlet as a Helmholtz resonator and exhaust as a wave tube.

![Figure 2-6: Frequency comparison between computational and experimental results.](image)

- a) Frequency vs. inlet diameter for 50 cm exhaust duct pulsejet;
- b) Frequency vs. exhaust duct length for 1.6 cm inlet diameter.
Table 2-1: Chamber temperature from simulations and experimental data

<table>
<thead>
<tr>
<th>Inlet diameter (cm)</th>
<th>Simulation temperature (K)</th>
<th>Experimental temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.3</td>
<td>1711</td>
<td>1593</td>
</tr>
<tr>
<td>1.6</td>
<td>2000</td>
<td>1423</td>
</tr>
<tr>
<td>2.2</td>
<td>1826</td>
<td>N/A</td>
</tr>
<tr>
<td>2.5</td>
<td>1807</td>
<td>1343</td>
</tr>
</tbody>
</table>

As seen from Figure 2-6, the frequencies obtained from simulations are 5–10% higher than those from experiments. These disagreements are explained by considering the temperature difference between experiments and simulations. The higher the temperature, the higher the speed of sound and thus the higher the frequency. As shown in Table 2-1 (unfortunately, experimental temperature for 2.2 cm inlet diameter pulsejet was not acquired), it was found that the average temperature from the simulations at Port 3 was much higher than measured. There are two reasons for this. First, in the two-dimensional simulations, one source point was set in a 4 deg wedge, which equates to 90 point fuel sources for the whole pulsejet; however, in the experiments, there were only eight small holes in the fuel injector. Since the reaction is controlled by fuel-air mixing, this led to a significantly shorter combustion time and thus higher temperature in the simulations than in the experiments. This is the result of the compromise between precision and simulation time expense. Second, there is always heat transfer from the junction of the thermal couple through the wire. At Port 3, this temperature difference is quite large, and the distance from the centerline to the water jacket is quite small. This causes lower experimental temperature reading than real temperature at Port 3. However, since temperature has the same effect on
inlet and exhaust frequencies, this difference will not affect on the inlet-exhaust frequency matching discussed (Eq. 2-3) in this paper.

Figure 2-7 compares the inlet frequency calculated by Eq. 2-2 and the exhaust frequency calculated by Eq. 2-1 with operating frequency obtained in either simulations or experiments. In Figure 2-7(a), the inlet and exhaust frequencies are calculated by using temperature from the computational results, and in Figure 2-7(b), these are calculated by using experimental temperature results. From these figures, it is clear that the frequencies obtained from both simulations and experiments compare well to the average frequency calculated by the two analytical acoustic models. Moreover, at small inlet diameters, the wave tube (exhaust) model overpredicts the operating frequency while the Helmholtz (inlet) model underpredicts the frequency. This then flips as the inlet diameter increases.

![Figure 2-7: Calculated inlet and exhaust frequency compared with pulsejet running frequency for 50 cm tail pipe. a) Computational data; b) Experimental data.](image-url)
Table 2-2 shows a comparison of calculated frequencies with the actual measured frequencies for all available experimental results. The inlet frequency was calculated by Eq. (2-2), the exhaust frequency was calculated by Eq. (2-1), and the overall operating frequency was calculated by our new acoustic model, Eq. (2-3). As shown, the new acoustic model frequencies compare very well to the measured frequencies, with the difference between them being less than 5% for all tested cases. Simulation results for the same geometries are summarized in Table 2-3. It is evident that frequencies obtained using the computational model compare well to frequencies obtained using the analytical model, the difference being less than 10% for all cases. This suggests that the new analytical acoustic model given by Eq. (2-3) captures the relevant physics and yields correct results.

Table 2-2: Frequencies obtained from experimental data versus those obtained from the analytical model

<table>
<thead>
<tr>
<th>Tail pipe length (cm)</th>
<th>Inlet length (cm)</th>
<th>Inlet diameter (cm)</th>
<th>Inlet frequency (Eqn. 2) (Hz)</th>
<th>Exhaust frequency (Eqn. 1) (Hz)</th>
<th>Acoustic model (Eqn. 3) (Hz)</th>
<th>Measured Jet Freq (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>42</td>
<td>10</td>
<td>1.6</td>
<td>208</td>
<td>254</td>
<td>231</td>
<td>220</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>247</td>
<td>258</td>
<td>253</td>
<td>253</td>
</tr>
<tr>
<td>50</td>
<td>10</td>
<td>1.3</td>
<td>164</td>
<td>212</td>
<td>188</td>
<td>183</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.6</td>
<td>213</td>
<td>214</td>
<td>214</td>
<td>214</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>275</td>
<td>211</td>
<td>243</td>
<td>244</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.5</td>
<td>291</td>
<td>216</td>
<td>253</td>
<td>259</td>
</tr>
<tr>
<td>58</td>
<td>10</td>
<td>1.3</td>
<td>158</td>
<td>181</td>
<td>169</td>
<td>178</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.6</td>
<td>212</td>
<td>183</td>
<td>198</td>
<td>196</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>275</td>
<td>176</td>
<td>226</td>
<td>223</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.5</td>
<td>297</td>
<td>202</td>
<td>249</td>
<td>246</td>
</tr>
</tbody>
</table>

The frequencies obtained from the analytical acoustic model depend on both inlet and exhaust temperatures through the speed of sound. Average temperatures at the inlet and
exhaust provide some information about the relative amount of time during each cycle when the air flows into the pulsejet (i.e., at 300 K), from both ends, and when combustion products flow out (i.e., ~1500 +K), also from both ends.

Table 2-3: Frequencies obtained from a computer simulation versus those obtained from the analytical model

<table>
<thead>
<tr>
<th>Tail pipe length (cm)</th>
<th>Inlet length (cm)</th>
<th>Inlet diameter (cm)</th>
<th>Inlet frequency (Eqn. 2) (Hz)</th>
<th>Exhaust frequency (Eqn. 1) (Hz)</th>
<th>Acoustic model (Eqn. 3) (Hz)</th>
<th>Simulation Jet Freq (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>42</td>
<td>10</td>
<td>1.6</td>
<td>222</td>
<td>238</td>
<td>230</td>
<td>244</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>323</td>
<td>255</td>
<td>289</td>
<td>265</td>
</tr>
<tr>
<td>7</td>
<td>1.3</td>
<td>212</td>
<td>208</td>
<td>210</td>
<td>218</td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>10</td>
<td>1.3</td>
<td>181</td>
<td>193</td>
<td>187</td>
<td>196</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.6</td>
<td>223</td>
<td>214</td>
<td>219</td>
<td>227</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>298</td>
<td>222</td>
<td>260</td>
<td>258</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.5</td>
<td>306</td>
<td>231</td>
<td>269</td>
<td>271</td>
</tr>
<tr>
<td>7</td>
<td>1.3</td>
<td>211</td>
<td>184</td>
<td>198</td>
<td>208</td>
<td></td>
</tr>
<tr>
<td>58</td>
<td>10</td>
<td>1.3</td>
<td>171</td>
<td>179</td>
<td>175</td>
<td>190</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.6</td>
<td>227</td>
<td>189</td>
<td>208</td>
<td>210</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.2</td>
<td>316</td>
<td>196</td>
<td>256</td>
<td>250</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.5</td>
<td>300</td>
<td>214</td>
<td>257</td>
<td>242</td>
</tr>
</tbody>
</table>

As shown in Figure 2-8, for a small inlet diameter, the average temperature of the fluid moving through the pulsejet at the inlet is higher than at the exhaust. As the inlet diameter increases, the average inlet temperature decreases while the exhaust average temperature increases. Computations and experiments agree quite well; however, this result is counterintuitive, as one would expect a small diameter inlet to force more hot products out the exhaust.

To validate the external flow field simulation, computed temperatures at the exit and at several points downstream are compared to measured temperatures. As shown in Figure 2-9,
simulation and experimental results compare reasonably well. The numerical model is thus validated by comparing with experimental data for the temperature (both inside and outside the pulsejet) and pressure at various locations, as well as for the operating frequency.

![Figure 2-8: Inlet and exit temperatures vs. the inlet diameter for a 50 cm exhaust duct pulsejet](image)

![Figure 2-9: Temperature at the exit](image)
Thrust was not experimentally measured, but was calculated based on the momentum flux through the inlet and exhaust. Thrust generated by the momentum flux through the inlet and exhaust planes was separately calculated, and the total thrust was calculated by deducting the inlet thrust from the exhaust thrust. Studying the dependence of the total thrust on the inlet diameter suggests that the maximum thrust occurs for the inlet diameter at which the inlet and exhaust frequencies coincide. To confirm this assumption, a new case with the inlet diameter determined by the intersection of the curves corresponding to inlet and exhaust frequencies in Figure 2-7(a) was computationally investigated to obtain the thrust for the geometry where the inlet and exhaust frequencies coincide.

For the base line exhaust duct length of 50 cm, the inlet thrust, exhaust thrust, and net thrust were calculated as a function of inlet diameter and the results are shown in Figure 2-10. The inlet thrust (in the negative $x$ direction) increases with inlet diameter. The exhaust thrust
initially increases, is reasonably constant, and then drops as the inlet diameter increases. The net thrust (exhaust thrust minus inlet thrust) is observed to peak at an inlet diameter corresponding to the inlet-exhaust frequency matching condition. It is also clear that as the inlet diameter gets large, the net thrust becomes negative, as expected.

The same simulations were also carried out for 42 cm and 58 cm exhaust duct lengths. The results are displayed in Figure 2-11. The pulsejet with 1.70 cm inlet diameter and 42 cm exhaust duct length and 1.32 cm inlet diameter with the 58 cm exhaust duct length generate the greatest net thrust. These are also the inlet diameters, which make the inlet frequencies coincide with exhaust frequencies.

Figure 2-11: Thrust vs. inlet diameter

(a) 42 cm exhaust duct pulsejet; (b) 58 cm exhaust duct pulsejet

Figure 2-10 and Figure 2-11 also show that there is a strong correlation between the peak pressure in the combustion chamber and the matching condition of inlet and exhaust
frequencies. The correlation is most obvious for a pulsejet with a shorter exhaust duct. For a 42 cm exhaust duct pulsejet (see Figure 2-11(a)), the peak combustion chamber pressure takes on a maximum value when the inlet and exhaust frequencies are coincident, and is less sensitive at longer lengths. As expected, the net thrust is correlated with peak chamber pressure.

From the inlet frequency equation (Eq. 2-2), it is clear that another way to change the inlet frequency is to change the inlet length. For a pulsejet with a 58 cm exhaust duct length and inlet diameter of 1.6 cm, the inlet-exhaust frequency match condition yields a length of 14 cm. Simulations were performed for this pulsejet as well as for inlet lengths of 10 cm and 17 cm. Results are shown in Figure 2-12. Again, the greatest net thrust is obtained for the case when the inlet frequency is equal to exhaust frequency.

![Graph](image)

*Figure 2-12: Thrust vs. inlet length for 58 cm exhaust duct pulsejet*
These results lead to a conclusion that the total thrust of a valveless pulsejet is maximized when the inlet frequency is equal to the exhaust frequency. This is explained by the resonance between the inlet and exit ducts that enhances waves traveling through the pulsejet, thus improving the pulsejet performance.

![Figure 2-13: Chamber pressure 2.54 cm inlet diameter, 0.42 m exhaust duct pulsejet.](image)

Table 2-4: Simulation results for smallest and largest diameters for 42 cm exhaust duct pulsejet

<table>
<thead>
<tr>
<th>Inlet diameter (cm)</th>
<th>Inlet frequency (Hz)</th>
<th>Exhaust frequency (Hz)</th>
<th>Inlet frequency / Exhaust frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.27</td>
<td>168</td>
<td>215</td>
<td>0.78</td>
</tr>
<tr>
<td>2.22</td>
<td>323</td>
<td>255</td>
<td>1.27</td>
</tr>
</tbody>
</table>

The obtained results also suggest that when the difference between the inlet and exhaust frequencies exceeds a certain critical value, pulsejet operation becomes impossible. Table 2-4 gives the smallest and largest inlet diameters of the pulsejet for which sustainable operation is observed. Although these values do not provide exact boundaries of the region when pulsejet operation is possible, they reflect the relationship between the frequency and operation possibility. Figure 2-13 shows the simulated chamber pressure for a pulsejet with
2.54 cm inlet diameter and 42 cm exhaust duct length. The inlet frequency increases with the inlet diameter and in this case, it exceeds a tolerable inlet and exhaust frequency difference. As shown in Figure 2-13, the amplitude of the pressure oscillation in the pulsejet decays and operation ceases.

2.5.2 Pulsejet in Convective Stream

Simulations were also performed for pulsejets in a freestream. Results are shown for a 42 cm exhaust duct pulsejet with 1.0–2.2 cm inlet diameter and 0.47 g/ s fuel flow rate at 0 (static) m/s, 30 m/ s, 50 m/ s, and 80 m/ s freestream velocities in Figure 2-14. As seen in this figure, as the inlet diameter increases, the net thrust initially slightly increases before dramatically falling off. Also notable is that as the free stream velocity increases, the inlet diameter corresponding to the maximum net thrust decreases.

![Figure 2-14: Thrust vs. inlet diameter for different free stream speeds](image)
To determine the inlet diameter yielding the maximum net thrust, inlet and exhaust frequencies were compared. Simulation results suggest that the higher the freestream velocity, the larger the difference between the inlet diameters for the maximum net thrust and the inlet diameter that makes the inlet and exhaust frequencies equal.

![Figure 2-15: Thrust vs. inlet diameter. (a) 30 m/s free stream; (b) 120 m/s free stream](image)

![Table 2-5: a get from 50 and 80 m/s results](table)

<table>
<thead>
<tr>
<th>Free stream speed (m/s)</th>
<th>Values near peak Thrust</th>
<th>In (Exhaust / Inlet Frequency)</th>
<th>ln(1-M)</th>
<th>a</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>164</td>
<td>213</td>
<td>0.261</td>
<td>-0.1556</td>
</tr>
<tr>
<td>80</td>
<td>154</td>
<td>230</td>
<td>0.401</td>
<td>-0.2620</td>
</tr>
</tbody>
</table>

Since pulsejet frequencies are determined by acoustic waves traveling through the pipe, an explanation of this phenomenon is that the freestream velocity changes the inlet frequency. It is expected that this change is related to the Mach number $M$ of the freestream. It is
hypothesized that the freestream velocity increases the inlet frequency by the factor of \((1 - M)^a\). Table 2-5 shows the simulation results used to evaluate the power \(a\) by using the data for 50 m/s and 80 m/s convective streams. These simulations suggest that \(a\) is equal to approximately \(-1.5\). Equation 2-2 is then modified to account for the inlet flow velocity as follows:

\[
 f_i = \left[ \frac{c_i}{2\pi} \right] \left[ \frac{S_i}{VL_i} \right]^{0.5} (1 - M)^{-1.5}
\]  

(2-4)

This new equation for \(f_i\) is used to determine the inlet diameter for the maximum thrust for the inlet flow velocities of 30 m/s and 120 m/s. The predicted inlet diameters for these two cases are 1.5 cm and 1.1 cm, respectively. Figure 2-15 shows thrust versus inlet diameter for the inlet flow velocities of 30 m/s and 120 m/s. The maximum net thrust is observed to exactly occur at the diameter predicted by Eq. (2-4) in each case.

2.6 Conclusions

From the coupled experimental and computational investigation, the following conclusions are obtained.

1. The operating frequency of the valveless pulsejet depends on the average sound speed (dictated by temperature) and the geometry of both the inlet and exhaust. The average of the frequencies calculated with the one-sixth wavelength equation for the exhaust duct and the Helmholtz resonator equation for the inlet tube predicts the operating frequency of the pulsejets observed in both simulations and experiments within an
uncertainty of 10%. The operating frequency increases when the inlet diameter increases, the inlet length decreases, and when the exhaust duct length decreases.

2. The two-dimensional model simulates the experiment reasonably well. Different fuel supply configurations result in the chamber temperature and frequency in the simulation being 5–10% higher than in the experiment.

3. Increasing inlet diameter results in decreasing average inlet temperature, increasing average exhaust temperature, and increasing net mass flow traveling through the pulsejet. The decrease in average inlet temperature suggests that fewer combustion products leave through the inlet with increasing inlet diameter, contrary to intuition.

4. A new analytical acoustic model, where the inlet is treated as a Helmholtz resonator and the exhaust duct is treated as a wave tube, is shown to accurately predict the operating frequency.

5. The difference between inlet (Helmholtz) and exhaust (wave tube) frequencies has a great effect on the operation of a pulsejet. When the inlet frequency is equal to the exhaust frequency, the maximum net thrust is obtained. Conversely, if the difference between the inlet and exhaust frequencies exceeds a critical value, pulsejet operation ceases.

6. A freestream significantly changes the operation of the pulsejet. As the freestream velocity increases, the peak thrust occurs at smaller inlet diameters (or longer inlet lengths). This smaller inlet diameter is predicted by taking into account the freestream Mach number, and modifying the inlet frequency by the factor of 

\[(1-M)^{-1.5}\].
REFERENCES


CHAPTER 3. THE EFFECT OF INLET SIZE AND INTERIOR OBSTRUCTIONS ON THE PERFORMANCE OF VALVELESS PULSEJETS

In this chapter, computational methods are used to investigate how modification of the inlet size and placing an obstruction in the combustion chamber affect the overall performance of a valveless pulsejet in both a static and convective free stream. Different inlet lengths and diameters are simulated to optimize the geometry to yield largest net thrust. A 0.12 volume ratio between the inlet and combustion chamber is shown to be the best for a static pulsejet. However, this ratio must be decreased as the free stream velocity increases to account for ram compression. Different interior obstructions are also placed in the combustion chamber to modify the flow pattern and increase thrust. Results indicate that a properly placed obstruction in the pulsejet can significantly improve its performance.

3.1 Introduction

The pulsejet is an unsteady propulsion device that generates intermittent thrust [1]. The reactants enter the combustion chamber when the pressure in the combustion chamber is lower than the ambient pressure. Residual hot gases and heat transfer from the hot walls raise the reactants’ temperature to the auto-ignition temperature, initiating combustion of the reactants. The ensuing heat release increases pressure, and the combustion gases then expand down the exhaust duct and exit at high velocity, generating thrust. Once the combustion gases at the exit have expanded to nearly atmospheric pressure, the
overexpansion due to momentum of the combustion products causes the pressure to decrease in the combustion chamber to sub-atmospheric causing injection of fresh reactants. The pulsejet is also an acoustically resonant device and the acoustic wave make four trips during this time. The arrival of the compression acoustic wave corresponds with the heat release and the cycle repeats itself.

In traditional valved pulsejets, reed valves allow the injection of fresh reactants but prevent hot gases from leaving through the inlet. However, the use of reed valves limits the reliability and longevity of the engine, and renders the pulsejet difficult to scale down in size. In a valveless design, it is desirable to both eliminate the reed valves and minimize the mass flux leaving the inlet. By proper utilization of wave processes in an inlet duct of adequate length, the amount of negative thrust can be minimized in a valveless pulsejet [2].

In 1909, Georges Marconnet developed the first pulsating combustor without valves (Figure 1-2). The principle of the valveless pulsating combustor was rediscovered by Lt. William Schubert of the US Navy in the early 1940s. His design, called the “resojet” at the time on account of its dependence on resonance, is one of the simplest successful valveless designs (Figure 1-3) [3].

There were numerous designs of valveless pulsejets developed after the World War II [4-10], but they were put aside for decades after that due to their relatively low efficiency compared with the rapidly advancing turbojet. Due to the proliferation of UAVs, there is considerable interest in small-scale combustion devices. However, because the efficiency of turbo
machinery decreases nonlinearly with its size [11], there may be applications where the pulsejet’s scalability, simplicity, and low-cost make it an attractive propulsion device.

In previous work [12, 13], a modified version of the BMS (Bailey Machining Service) hobby-scale (50 cm in length) pulsejet was studied where the original valved inlet is replaced with a straight pipe (Figure 2-1).

An acoustic model was developed for the valveless pulsejet. It treats the valveless pulsejet as two separate but coupled parts, the inlet and exhaust. The combination of the tailpipe and combustion chamber was modeled as a wave tube, the same way as in the previous investigation of a valved pulsejet [14]. Thus the exhaust frequency is calculated by:

\[
\frac{\omega_e V}{S_e c_e} \tan \frac{\omega_e L_e}{c_e} = 1
\]  

(3-1)

where \(\omega_e = 2\pi f_e\), \(f_e\) is the resonate frequency associated with the exhaust pipe, \(c_e\) is the average speed of sound in the exhaust tube, \(S_e\) is the cross sectional area of the exhaust pipe, \(V\) is the volume of the combustion chamber, and \(L_e\) is the length of the exhaust pipe.

On the other hand, the combination of the short inlet and the combustion chamber was modeled as a Helmholtz resonator because the volume of the combustion chamber is much larger than the volume of the inlet. Thus the inlet frequency is calculated by [15]:

\[
f_i = \left[ c_i / (2\pi) \right] \left[ S_i / (V L_i) \right]^{0.5}
\]

(3-2)
where $f_i$ is the resonate frequency associated with the inlet, $c_i$ is the average speed of sound in the inlet and combustion chamber, $S_i$ is the cross section area of the inlet, $V$ is the volume of the combustion chamber, and $L_i$ is the length of the inlet.

Both experimental and numerical results support the conclusion that the valveless pulsejet will only operate when the two frequencies are approximately equal, and the operating frequency of the pulsejet approximately equals the average of the inlet and exhaust frequencies [13]. Numerical results obtained for different inlet lengths and diameters also suggest that the best performance is observed when inlet and exhaust frequencies are the same.

A numerical study was also performed to model the pulsejet in a convective free stream. Different free stream velocities were simulated. Computational results suggested that the inlet frequency should be scaled by a factor of $(1 - M)^{-1.5}$ when the pulsejet is moving through an external flow field to account for the finite wave propagation time in the inlet:

$$f_i = \frac{c_i}{(2\pi)}\left[\frac{S_i}{V/L_i}\right]^{0.5}(1 - M)^{-1.5} \quad (3-3)$$

where $M$ is the free stream Mach number.

### 3.2 Numerical model

The commercially available CFD package CFX™ 5.7 is used to model the combustion and flow inside and outside of the pulsejet. The computation model geometry is exactly the same as the experimental model, as shown in Figure 2-1. Because the pulsejet is symmetric about
its axis, the two-dimensional axisymmetric formulation of the problem is utilized to save computational time; the computational domain is shown in Figure 2-3. Much smaller mesh size is used for the flow field inside the pulsejet than for the flow field outside the pulsejet. The total number of nodes in this model is about 18,000. Simulations were also performed for models with twice the mesh densities, and the results have shown that further increasing of the node density does not affect the results. Computations were performed on the NC State IBM Blade Center utilizing a single 3.0 GHz Inter Xeon processor. Typical computational time for one cycle of the pulsejet was about 18 CPU hours.

A k-ε model based on the Reynolds Averaged Navier-Stokes (RANS) equations was used because it offers a good compromise in terms of accuracy and robustness [16, 17]. The Eddy Dissipation Model was chosen as the combustion model. A propane-air five step reaction mechanism provided by CFX 5.7 was used to simulate the combustion processes, which include reactions for propane oxidation, hydrogen oxidation, CO oxidation and both directions of the water-gas shift reaction.

As shown in Figure 2-3, the computational domain included not only the interior of the pulsejet but also an extended domain to simulate the flow field around the pulsejet. Because the boundaries were set so far from the pulsejet, effects from the pulsejet operation were negligible at the boundaries; the boundary condition was set to be 300 K temperature and 10^5 Pa pressure.
There is a significant heat flux between the pulsejet walls and the fluid inside the pulsejet. However, the frequency is high enough and the thermal inertia is large enough so that the wall temperature distribution can be assumed to be steady. This temperature distribution was calculated by a steady state simulation that was performed based on the experimentally measured average gas temperature. This was further simplified to what is shown in Figure 2-4, assuming a constant temperature of 1000 K along the combustion chamber and then an exponential decay toward both the inlet and exit planes at 400 K. Pulsejets running at different forward flight speeds were also modeled by changing the boundary condition at the inlet of the enclosure flow field.

The model described above is thus created by carefully choosing the fluid and combustion models to simulate the pulsejet; it is validated by comparisons with experimental results as explained in CHAPTER 2, where pressure, frequencies and temperatures are compared between experimental results and the numerical predictions for exactly the same pulsejet geometries.

3.3 Results and discussion

From the conclusion drawn in CHAPTER 2, the inlet frequency should match the exhaust frequency to generate the maximum thrust. However, from Eqn. 3-2, there is infinite number of inlet diameter and inlet length combinations to generate the same frequency to match the exhaust frequency. To find the best inlet geometry, valveless pulsejets (as shown in Figure 2-1) with 15.2 cm extension length were simulated. For this combustion chamber and tailpipe
geometry, by applying Eqn. 3-1 and 3-2, and using the temperature data from the data in CHAPTER 2, the matching frequency should be 220 Hz and the inlet geometry should meet the following condition:

\[
\frac{L_i}{S_i} = 5.35 \text{ cm}^1
\]  

(3-4)

where \(L_i\) is the length of the inlet and \(S_i\) is the cross section area of the inlet.

![Figure 3-1: Thrust and peak chamber pressure for different inlet diameter](image)

A series of inlet lengths and diameters satisfying Eqn. 3-4 were simulated. As the ratio of inlet and combustion chamber volume increases from 0.027 to 0.175, the operation frequency decreases from 220 Hz to 212 Hz. This is because that as the inlet volume increases, the Helmholtz relation (Eqn. 3-2), which is derived by assuming that the inlet volume is much smaller than the combustion chamber volume, losses its applicability. However, since the change of the frequency is small, one can say that the inlet frequency matches the exhaust
frequency for the selected geometries. The pressure and thrust results are shown in Figure 3-1. From this figure, the inlet volume variation has a significant effect on the performance of valveless pulsejets. The thrust generated by the pulsejet reaches a maximum when the inlet diameter is 1.9 cm, which is consistent with the peak combustion chamber pressure. This means that the ratio of the inlet and combustion chamber volumes should be about 0.12 to achieve the best performance.

Table 3-1: Total force of pulsejet with different inlet geometry in 30 m/s free stream

<table>
<thead>
<tr>
<th>Inlet diameter (cm)</th>
<th>Inlet length (cm)</th>
<th>Total force (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.59</td>
<td>10.16</td>
<td>0.46</td>
</tr>
<tr>
<td>1.65</td>
<td>10.92</td>
<td>0.93</td>
</tr>
<tr>
<td>1.73</td>
<td>11.94</td>
<td>1.15</td>
</tr>
<tr>
<td>1.78</td>
<td>12.70</td>
<td>1.29</td>
</tr>
<tr>
<td>1.85</td>
<td>13.72</td>
<td>1.00</td>
</tr>
<tr>
<td>2.03</td>
<td>14.99</td>
<td>0.65</td>
</tr>
</tbody>
</table>

Table 3-2: Total force of pulsejet with different inlet geometry in 50 m/s free stream

<table>
<thead>
<tr>
<th>Inlet diameter (cm)</th>
<th>Inlet length (cm)</th>
<th>Total force (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.40</td>
<td>10.67</td>
<td>-0.11</td>
</tr>
<tr>
<td>1.52</td>
<td>12.19</td>
<td>0.84</td>
</tr>
<tr>
<td>1.59</td>
<td>13.21</td>
<td>0.25</td>
</tr>
<tr>
<td>1.65</td>
<td>14.22</td>
<td>0.37</td>
</tr>
</tbody>
</table>

Simulations were also performed for the cases with a free stream velocity. Using Eqn. 3-3 rather than Eqn. 3-2, various combinations of inlet diameters and lengths were investigated to find the geometry yielding the best performance. Total force on the solid structure, which is equal to the thrust generated by the pulsejet minus the drag force caused by the free stream...
velocity, was used to evaluate the performance. Results from these simulations are shown in Table 3-1 (30 m/s) and Table 3-2 (50 m/s).

There is also a significant effect of inlet geometry on thrust for cases with a free stream around the pulsejet. However, the ratio of the inlet and combustion chamber volumes for largest thrust decreases to 0.1 and 0.07 for 30 m/s and 50 m/s free stream velocity, respectively. To decrease the drag force, a more aerodynamic outer geometry is used for the cases of free stream velocity greater than 50 m/s, as shown in Figure 3-2 (no difference is calculated between the original blunt geometry and the aerodynamic geometry at velocities less than 50 m/s).

![Figure 3-2: Oxygen mass fraction in the pulsejet in an 80 m/s free stream environment with aerodynamic outer geometry.](image)

Computational results for the pulsejet running in an 80 m/s flow field are shown in Figure 3-2, where the maximum O$_2$ penetration is depicted. It is evident that when the free stream velocity increases above a critical value, the air entering the inlet of the pulsejet convects all the way through the combustion chamber and enters the tailpipe without participating in the combustion process. The fuel is eventually consumed, but not in the combustion chamber,
resulting in high pressure in the tailpipe instead of in the combustion chamber. That is not what is desired, because, as shown in Figure 3-1, the performance of a valveless pulsejet is highly dependent on being able to generate elevated combustion chamber pressures, with higher pressure yielding larger net thrust.

An obstruction was thus placed at the throat of the pulsejet, as shown in Figure 3-3, to prevent inlet air from convecting through the combustion chamber and entering the tailpipe directly. Simulation results predict that an object at the throat not only prevents the air from passing through the combustion chamber, but also improves mixing in the combustion chamber. For a valveless pulsejet with 15.2 cm inlet length and 1.5 cm inlet diameter in a flow field with 80 m/s free stream velocity, the total force increases from 0.61 N to 1.6 N by adding an obstruction at the throat.

![Obstruction](image)

*Figure 3-3: Velocity vector in the pulsejet with obstruction placed at the throat.*

*Free stream velocity = 80 m/s*

Another way to prevent inlet air from just flowing through the combustion chamber is by placing an obstruction at the exit of inlet, as shown in Figure 3-4. Vortexes, which greatly improve the mixing and combustion process in the chamber, thus improving the overall performance, are generated just behind the bluff body obstruction. Note that the inner surfaces at the transition between the inlet and combustion chamber have been rounded to
smooth this transition and accommodate the obstruction. With the same inlet and tailpipe geometry as that for the case shown in Figure 3-3, the total force in a flow field with 80 m/s free stream velocity is predicted to be 1 N, still an improvement, but not as significant as with the obstruction at the throat.

The operating frequencies for these two cases are both 215 Hz, the same as for the case without an obstruction in the combustion chamber. According to the frequency relationship, the operation frequency is equal to the average of inlet frequency and exhaust frequency [13]. This indicates that adding interior obstructions do not affect inlet or exhaust frequency as long as they do not change the chamber volume appreciably. Thus, interior obstructions do not destroy the inlet and exhaust frequency matching, and can be designed separately from the inlet and tailpipe geometry.

![Figure 3-4: Velocity vector in the pulsejet with obstruction placed at the inlet-chamber transition](image)

To determine if there was a detrimental effect of having these interior obstructions in the combustion chamber at low free stream velocities, simulations were performed for both obstructions with a static free stream. Results indicate very little effect on pressure or thrust with obstruction compared with simulations without an obstruction.
3.4 Conclusions

From the analysis presented above, the following conclusions are obtained:

1. The inlet volume has a great effect on the performance of a valveless pulsejet. For a pulsejet operating in a static environment, the optimal ratio of inlet and chamber volumes is approximately 0.12 providing the best performance. This optimal ratio decreases as free stream velocity increases.

2. For a pulsejet running in the free stream, air from inlet can convect into the combustion chamber and enter the tailpipe without reacting in the combustion chamber at sufficiently high free stream velocities. Properly placed interior obstructions prevent this and improve the performance significantly.

3. Adding an interior obstruction does not affect the inlet and exhaust frequency, and could be designed separately from inlet and tailpipe geometry. There also is no penalty for having this obstruction at low free stream velocities.
REFERENCES


CHAPTER 4. SUMMARY OF THE PRELIMINARY WORK ON MODELING THE HELMHOLTZ PULSEJET ENGINE

4.1 Introduction

The reed valve of the traditional pulsejet is easy to be damaged in high temperature and high frequency environment, which limited the longevity of the valved pulsejet. The valveless pulsejet, without any moving parts, could achieve longer operation time. However, the shortcoming of a valveless pulsejet is that the hot gas leave from the inlet and generates negative thrust. One way to prevent hot gas from leaving from the inlet is to use the exit pipe as inlet. This idea could be dated back to 1961, in the collected work of Reynst [1]. In his work, Reynst make a jar with proper sized hole on it working as a pulsejet. This kind of jet could be easily made in home with a jam jar, and called “Jam Jar Jet” as toy.

Another motivation for develop a Helmholtz pulsejet is to provide a sound source to mimic the sound wave from Helicopter. Great noise generated by pulsed combustion is one of the shortcomings of pulsejet. However, it could also be used as an advantage under certain environment by adjusting its frequency to desired value.

In previous work on pulsejet modeling, it showed that the traditional pulsejet can be modeled as a 1/6 wave tube [2], whose operating frequency is calculated based on its length:

\[ f_0 = \frac{c}{6L_p} \]  (4-1)
where \( c \) is the speed of sound and \( L_p \) is the length of the pulsejet.

Different from a traditional wave tube, a Helmholtz pulsejet is modeled as a Helmholtz resonator.

A Helmholtz resonator or Helmholtz oscillator is a container of gas (usually air) with an open hole (or neck or port) (Figure 4-1). A volume of air in and near the open hole vibrates because of compressibility of the air inside.

\[
\text{Figure 4-1: Helmholtz resonator}
\]

As shown in Figure 4-1, while the volume of the neck is much smaller than that of the chamber, the mass of the air in the neck serves as a piston and the large volume of chamber serves as a spring.

The mass of air in the neck and the spring constant of the air in the chamber are given by the following expressions:

\[
m = \rho S L \\
\]

(4-2)

and

\[
K = \rho S^2 c^2 / V \\
\]

(4-3)
respectively, where \( \rho \) is the air density, \( c \) is speed of sound, \( S \) is the cross sectional area of the neck, \( L \) is the length of the neck, and \( V \) is the volume of the chamber, as shown in Figure 4-1.

The natural frequency of the Helmholtz resonator is given by [3]:

\[
 f_0 = \frac{1}{2\pi} \sqrt{\frac{K}{m}} = \frac{c}{2\pi} \sqrt{\frac{S}{VL}}
\]  

(4-4)

Unlike the traditional pulsejet, the natural frequency of the Helmholtz resonator is related not only on the length of the neck but also on the cross-sectional area and chamber volume. This makes it possible to run the Helmholtz pulsejet engine at a much lower frequency than the traditional pulsejet with the same length, which constitutes the major advantage of using the proposed Helmholtz pulsejet engine for emulating helicopter frequencies.

### 4.2 Numerical model

To study the feasibility and performance of a Helmholtz pulsejet engine, the commercially available CFD software package, CFX²⁷, is used to simulate combustion and fluid flow inside and outside a Helmholtz pulsejet.

Due to the similarity of the physical processes in the traditional and Helmholtz pulsejets, simulations of the Helmholtz pulsejet rely on the same combustion and flow models and the same initial and boundary conditions that we previously developed for traditional pulsejets. The validity of the model for traditional pulsejet engines is validated by extensive comparison with experimental data and recorded in our publications [4-6].
The model geometry for simulating the Helmholtz pulsejet engine is set up as follows. Because the pulsejet is symmetric about the axis, its geometry is simplified to two-dimensional axisymmetric to save computational time. The mesh is generated in a 4 degree slice instead of the full 360° region. As shown in Figure 4-2, the computational domain includes not only the interior of the pulsejet but also an extended domain to compute the flow field around the pulsejet. Because the remote boundaries are set so far from the pulsejet, effects from the pulsejet operation are negligible at these boundaries; at remote boundaries temperature is simply set to 300 K and pressure is set to $10^5$ Pa.

![Figure 4-2: Computational domain for modeling the Helmholtz pulsejet engine](image)

A much smaller mesh size is used for analyzing the flow field inside the pulsejet, as shown in Figure 4-3, than for the flow field outside the pulsejet. The total number of nodes in this model is about 5,000. The code is also tested for a double mesh density case. However, the results indicate that further increasing the density of nodes does not affect the results. The computations are performed on the NCSU IBM Blade Center utilizing a single 3.0 GHz Inter Xeon processor.
A \( k-\varepsilon \) model based on the Reynolds Averaged Navier-Stokes (RANS) equations is used because it offers a good compromise in terms of accuracy and robustness [7, 8]. The Eddy Dissipation model is chosen as the combustion model. A propane-air five step reaction mechanism (propane oxidation, hydrogen oxidation, CO oxidation and a water-gas shift reaction) provided by CFX 5 is used to simulate the combustion process.

There is a significant heat flux between the pulsejet walls and fluid inside the pulsejet. However, the frequency is high enough and the thermal inertia is large enough so that the wall temperature distribution can be assumed steady with a good degree of approximation.

To find the steady-state temperature distribution, a steady-state simulation is performed, where boundary conditions are based on the average gas temperature in a transient simulation. This temperature distribution is further simplified to what is shown in Figure 4-4, where the dashed lines represent the junction of the combustion chamber and tailpipe, with a
constant temperature of 1000 K along the combustion chamber and then an exponential
decay toward the exit plane, which is assumed to be at 400 K.

![Figure 4-4: Temperature distribution along the pulsejet wall](image)

Pressures, temperatures and velocities are monitored during the simulations at several points
in the pulsejet. Port 1 is set in the middle of the combustion chamber, port 3 is set at the
junction of the combustion chamber and the tailpipe, and port 5 is set at the exit plane, as
shown in Figure 4-2. Fuel is added into the system at port 1 at a constant flow rate. Propane
is chosen as fuel in all of presented simulations.

### 4.3 Results and discussion

#### 4.3.1 2 m Helmholtz pulsejet

The first try of the simulation of Helmholtz pulsejet is performed for a 2 meter long
Helmholtz resonator. The pulsejet is composed of a combustion chamber and a tailpipe
(Figure 4-5). The total length of the pulsejet is 2 m, the inner chamber diameter is 0.8 m, the
tailpipe length is 1.15 m.

The simulations indicate a stable operation of this pulsejet engine. Figure 4-5 presents a
snapshot of the temperature field in the cross-section of the pulsejet taken along its axis when
the fresh air enters into the combustion chamber through the tailpipe. White arrows show the vortex generated in the combustion chamber. Figure 4-6 shows the pressure distributions in ports 1, 3, and 5 of this pulsejet, the operating frequency is 14 Hz, exactly as predicted by the Helmholtz frequency equation, equation (4-1). It is much lower than the frequency of a traditional pulsejet which would be 50 Hz if calculated as a frequency of a 1/6 wave tube.

Figure 4-5: Air enters the combustion chamber of the 2 m Helmholtz pulsejet with 10.4 cm tailpipe diameter

Figure 4-6: Pressure profile for the 2 m Helmholtz pulsejet with 10.4 cm tailpipe diameter
This Helmholtz pulsejet generates 43.8 N thrust by consume 9 g/s propane fed into the jet continuously. The ISP is calculated to be 497 s, 27% larger than 391 s of a typical 50 cm valveless pulsejet.

A few different tailpipe diameters are used to evaluate the effect of the tailpipe diameter on the Helmholtz pulsejet. An interesting phenomenon is that for all of the tested tailpipe diameters, the average inhale velocity in the tailpipe (i.e. the average velocity over the tailpipe cross-section during the half cycle when fresh air enters into the pulsejet) is almost independent of the tailpipe diameter (Table 4-1).

<table>
<thead>
<tr>
<th>Tailpipe diameter (cm)</th>
<th>Inhale time duration (ms)</th>
<th>Average inhale velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>38.9</td>
<td>57.07</td>
</tr>
<tr>
<td>10.4</td>
<td>36.2</td>
<td>56.12</td>
</tr>
<tr>
<td>11</td>
<td>35.2</td>
<td>56.77</td>
</tr>
</tbody>
</table>

Table 4-1: Average inhale velocity for 2 m Helmholtz pulsejet with different tailpipe diameters

This makes it possible to find the best tailpipe diameter that would make it possible to maximize the amount of fresh air that goes into the combustion chamber. During the inhaling half of the cycle, air enters the pulsejet from its exit, passes through the tailpipe and goes into the combustion chamber. However, the air remaining in the tailpipe does not take part in the combustion; it is pushed out during the exhaling half of the cycle. Only the fresh air that enters the combustion chamber is useful for the combustion process. The volume of the fresh air that enters the combustion chamber during the inhaling half of the cycle is calculated as:

\[ Q = S(\bar{v}t - L) = \pi r^2(\bar{v}t - L) \]  

(4-5)
where \( Q \) is the volume of the fresh air, \( S \) is the cross-sectional area of the tailpipe, \( r \) is the radius of the tailpipe, \( L \) is the length of the tailpipe, \( \bar{v} \) is the average flow velocity in the tailpipe, and \( t \) is the duration of the inhaling half of the cycle.

Using Helmholtz equation (4-4):

\[
t = \frac{1}{2 f_0} = \frac{2\pi}{c} \sqrt{\frac{VL}{S}}
\]  

(4-6)

If \( V, L, \) and \( c \) are constant, differentiating \( Q \) with respect to \( r \) and setting this derivative to be zero gives the radius of the tailpipe that maximizes the intake of fresh air:

\[
r = \frac{\bar{v}}{2c} \sqrt{\frac{\pi V}{L}}
\]  

(4-7)

![Figure 4-7: Chamber pressure for the 2 m Helmholtz pulsejet with different tailpipe diameters](image)

For the pulsejet geometry described above, the optimal tailpipe diameter is calculated to be 10.4 cm. Pressure distributions for three different tailpipe diameters are shown in Figure 4-7.
The peak chamber pressure for the optimal tailpipe diameter (1.18 atm) is much higher than that for 11 cm diameter tailpipe (1.12 atm) and 9 cm diameter tailpipe (1.14 atm).

### 4.3.2 50 cm Helmholtz pulsejet

To compare with traditional pulsejets, 50 cm Helmholtz pulsejets were studied. The combustion chamber is made to be a cylinder with 25 cm diameter and 25 cm in length. The tailpipe is thus 25 cm to get a total length of 50 cm. Diameter of the tailpipe is calculated to be 2.3 cm by using equation (4-7).

Numerical simulations predicted the operating frequency of 35 Hz, as expected from Eq. (4-1). In the first run with 0.9 g/s fuel flow rate, the average net oxygen mass flow rate during the cycle is computed to be 0.72 g/s. Thus, to match the oxygen and fuel supply rates to get the stoichiometric condition, 0.2 g/s fuel flow rate is selected and used in all following simulations. Figure 4-8 shows snapshots at 4 different moments within a single cycle. These moments are defined as follows:

a) The chamber pressure equals to the atmospheric pressure and increasing;

b) The chamber pressure reaches its peak;

c) The chamber pressure drops to the atmospheric pressure and continues to decrease and

d) The chamber pressure reaches the lowest level.
From these figures, it is evident that a large eddy is formed in the combustion chamber; this eddy is quite stable and does not change significantly during the cycle. Because of this eddy, despite the fact that port 2 is very close to the tailpipe exit, the velocity at the location of port 2 is never directed toward the exit. This feature makes it safe to add fuel at the location of port 2 without letting the fuel escape to the tailpipe exit. Since the combustion benefits from
maximizing mixing between the fuel and fresh air, the fuel injection point is moved from port 1 to port 2 in all of our simulations for the 50 cm Helmholtz pulsejet.

Figure 4-9: Chamber pressure of the 50 cm Helmholtz pulsejet

Figure 4-9 shows the combustion chamber pressure for the 50 cm Helmholtz pulsejet engine (the temperate filed for this pulsejet at different moments is displayed in Figure 4-8). Due to a small volume ratio between the tailpipe and combustion chamber, the peak pressure for the 50 cm Helmholtz pulsejet is much smaller than that for the 2 m Helmholtz pulsejet.

However, this can be improved by placing an object in the combustion chamber to increase turbulence in the chamber. Computational results with two circular rings placed in the combustion chamber are shown in Figure 4-10; these objects are placed near the end of the combustion chamber, so that mixing is locally improved in this region due to enhanced turbulence. The results indicate that mixing is indeed enhanced and the peak chamber pressure is increased by 38%. This result is shown in Figure 4-11.
Another way to improve the performance is to increase the volume ratio between tailpipe and combustion chamber. Increase diameter along, however, is not applicable. Recall Helmholtz equation (4-4), increase cross-sectional area of the tailpipe will increase the operation frequency of the pulsejet and thus decrease fresh air taken into the combustion chamber. Thus a longer tailpipe is desired.

From equation (4-5), as long as \((\bar{v}t - L)\) increase with \(L\), or:

\[
\begin{align*}
9.80E+04 & \quad 1.00E+05 & \quad 1.02E+05 & \quad 1.04E+05 \\
0 & \quad 0.02 & \quad 0.04 & \quad 0.06 \\
\text{Pressure (Pa)} & \quad \text{Time (s)}
\end{align*}
\]
\[ \sqrt{\frac{\pi}{c}} \sqrt[\gamma]{V} < 1 \]  

(4-8)

Fresh air entering into the combustion chamber in a cycle will increase with the tailpipe length. Figure 4-12 shows the combustion chamber pressure for different tailpipe length. By double the tailpipe length from 25 cm to 50 cm, the peak gauge pressure in combustion chamber also doubled. The thrust for the 50 cm tailpipe Helmholtz pulsejet is calculated to be 0.34 N, with ISP increased to 173.

As shown in Table 4-2, due to low peak pressure, 50 cm Helmholtz pulsejets in current version generates much lower thrust compare to the same length valved and valveless pulsejet. However, it could provide much lower frequency with the same length. By either improve mixture in the combustion chamber or increase volume ratio between tailpipe and combustion chamber could greatly increase the performance of the Helmholtz pulsejet.

![Figure 4-12: Combustion chamber pressure for different tailpipe length](image)

68
Table 4-2: performance of 50 cm pulsejets in different types

<table>
<thead>
<tr>
<th></th>
<th>Peak pressure (bar)</th>
<th>Average thrust (N)</th>
<th>Fuel flow rate (g/s)</th>
<th>ISP (s)</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Typical Valved pulsejet</td>
<td>1.7</td>
<td>23</td>
<td>1.5</td>
<td>1565</td>
<td>213</td>
</tr>
<tr>
<td>Typical Valveless pulsejet</td>
<td>1.3</td>
<td>1.8</td>
<td>0.47</td>
<td>391</td>
<td>213</td>
</tr>
<tr>
<td>Helmholtz pulsejet</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25 cm tailpipe</td>
<td>1.025</td>
<td>0.12</td>
<td>0.2</td>
<td>61</td>
<td>35</td>
</tr>
<tr>
<td>With Object</td>
<td>1.04</td>
<td>0.2</td>
<td>0.2</td>
<td>102</td>
<td>35</td>
</tr>
<tr>
<td>50 cm tailpipe</td>
<td>1.05</td>
<td>0.34</td>
<td>0.2</td>
<td>173</td>
<td>23</td>
</tr>
</tbody>
</table>

4.4 Conclusions

From analysis presented above, the following conclusions are obtained:

1. Properly combined combustion chamber and tailpipe could be made to be a continuously running Helmholtz pulsejet. A stable large eddy will be formed in a running Helmholtz pulsejet.

2. The operational frequency of a Helmholtz pulsejet is much smaller than a traditional pulsejet in same length. The ISP of a 2 m Helmholtz pulsejet is comparable to a valveless pulsejet.

3. The volume ratio between tailpipe and combustion chamber will be decreased for a smaller Helmholtz pulsejet, which will decrease the peak pressure of the process.

4. By either improve mixture in the combustion chamber or increase volume ratio between tailpipe and combustion chamber could greatly increase the performance of the Helmholtz pulsejet.
REFERENCES


CHAPTER 5. USE MATLAB SIMULATION TO FAST EVALUATE PULSEJET PERFORMANCE

Nomenclature:

\( C_v \) Specific heat capacity at constant volume

\( H_{vf} \) reaction heat of fuel

\( m \) mass in combustion chamber

\( m_f \) mass of fuel in combustion chamber

\( P \) Average Pressure in combustion chamber

\( q \) Heat transfer rate

\( R \) gas constant

\( T \) Average Temperature in combustion chamber

\( u \) specific internal energy

\( v \) velocity

\( V \) chamber volume

\( \rho_c \) Average density in combustion chamber

\( \gamma \) specific heat capacity ratio
5.1 Introduction

Pulsejet is operated upon acoustic phenomenon. The acoustic wave travel back and forth in the pulsejet, achieve self compression in the combustion chamber. It is desired that the combustion process happens at high chamber pressure, which requires sufficient air intake, proper air intake time from the inlet and good mixing process in the combustion chamber.

In CHAPTER 2, valveless pulsejet is modeled as combination of a Helmholtz resonator (inlet) and a wave tube (exhaust tube). A valveless pulsejet need to have inlet frequency match with exhaust frequency to properly operate. However, there are infinite combinations of inlet diameter and inlet length to compatible with a given combustion chamber and tail pipe geometry. In CHAPTER 3, inlets with different lengths and diameters were simulated for a given combustion chamber volume and tailpipe, an optimal ratio of inlet and chamber volumes, approximately 0.12, is provided.

The simulation with CFX in previous work is precision enough to predict the performance of valveless pulsejet, but for prediction purpose, the computational time sometimes is too long to be affordable. For a Two-Dimensional (2-D) simulation of valveless pulsejet, the typical simulation time is 18 hours per cycle in our HPC system with single 3.0 GHz Inter Xeon processor. To get a stable simulation result, 3 cycles is needed. There are basically 5 parameters to decide the acoustic feature of a valveless pulsejet (inlet length, inlet diameter, combustion chamber volume, tailpipe length and tailpipe diameter), from the equations in
CHAPTER 2. Thus, thousands simulations and years computation time are needed to get a set of approximately optimal value.

The purpose of this paper is to provide a less precise but much faster solution. Although not precise enough, but this method could offer a trend of how the geometries effect on the pulsejet performance. Results could offer a much smaller range for optimal geometry candidates, and save time on 2-D simulations.

5.2 Physical model

The pulsejet process is very complex. However, it could be simplified as combinations of several simpler processes, combustion chamber heat addition, pipe flow, combustion chamber compression and expansion.

5.2.1 Chamber heat addition:

Combustion chamber pressure change could be divided to 2 separate processes, pressure change due to the inner energy change and pressure change due to fluid entering and leaving the control volume. These two processes could be considered separately during a sufficiently short period, and the combination of these processes simulates the real process during this period.

By doing this, the heat addition process could be assumed to be constant volume heat addition, since fluid entering or leaving the control volume will be considered separately. Assume thermally perfect gas:
\[ du = C_v dT = \frac{R}{\gamma - 1} dT = \frac{dP}{(\gamma - 1)\rho_c} \]  

(5-1)

Assume complete reaction:

\[ du = (dm_f \cdot H_{vf} + dq) / m \]  

(5-2)

Substitute equation (5-1) into (5-2):

\[ dp = \frac{(\gamma - 1)\rho_c (H_{vf} dm_f + dq)}{m} = \frac{\gamma - 1}{V} \left( H_{vf} \frac{dm_f}{dt} + \frac{dq}{dt} \right) dt \]  

(5-3)

If integrated in a sufficiently short period, the temperature in the combustion chamber did not change much, the \( \gamma \) value and reaction rate could be treated as constant:

\[ \Delta p = \int \frac{\gamma - 1}{V} (H_{vf} \dot{m}_f + \dot{q}) dt \equiv \frac{\gamma - 1}{V} (H_{vf} \dot{m}_f + \dot{q}) \Delta t \]  

(5-4)

Similarly, the temperature change is calculated as:

\[ \Delta T = \frac{T}{p} dp \approx \frac{(\gamma - 1)T}{pV} (H_{vf} \dot{m}_f + \dot{q}) \Delta t \]  

(5-5)

### 5.2.2 Mass flow into and out of combustion chamber

Another scheme to change the combustion chamber pressure is mass flow cross the boundary of the control volume of combustion chamber. Because the heat transfer process is separated from this process, isentropic expansion and compression is assumed:
\[ \frac{p}{\rho} = \text{const} \]  

(5-6)

Differential at both sides:

\[ dp = \gamma p \frac{d\rho}{\rho_c} \]  

(5-7)

Where \( \rho = \frac{m}{V}, \) \( d\rho = \frac{dm}{V} = \frac{\rho_c A v dt}{V}, \) substitute into (5-7),

\[ dp = \frac{\gamma p A v}{V} dt \]  

(5-8)

Again, if integrated in a sufficiently short period,

\[ \Delta p = \frac{\gamma p A v}{V} \Delta t \]  

(5-9)

Similarly,

\[ \Delta T = \frac{(\gamma - 1) T A v}{V} \Delta t \]  

(5-10)

5.2.3 Pipe flow simulation:

Pipe flow simulation is made by using 1-D CFD code programmed using Matlab language. The CFD code using cell centered finite volume method with given space-wise approximation over a control volume to solve the 2D compressible Euler equations. Van Leer’s type Flux splitting Scheme was used in this work. For robust purpose, 1\textsuperscript{st} order scheme is adopted.
Heat transfer and viscous loss in the tubes are also taken into account. A widely used method of simulating the wall type boundary condition is to use ghost volumes. However, to decrease the simulation time, in this work, heat transfer and viscous loss are calculated by experience equations [1, 2].

For laminar flow, friction factor was calculated as:

\[ f = \frac{64}{\text{Re}} \]  \hspace{1cm} (5-11a)

For turbulence flow, friction factor was calculated as:

\[ f = 0.316 \text{Re}^{-0.25} \]  \hspace{1cm} (5-11b)

Heat transfer is calculated as:

\[ \text{Nu} = 0.3 + 0.62 \text{Re}^{0.5} \left[ \frac{\text{Pr}^{1/3}}{1 + ((\gamma - 1)/\text{Pr})^{2/3}} \right]^{1/4} \left[ 1 + \left( \text{Re} \left( \frac{\text{Re}}{282000} \right)^{5/8} \right)^{0.8} \right] \]  \hspace{1cm} (5-12)

The boundary conditions at both ends of the pipe were set to be the opening type. The pressure and temperature in the combustion chamber were set as boundary conditions at one end and the atmospheric pressure and temperature were set as boundary conditions at the other end.

The simulation results of the flow in the pipes were then used to calculate combustion chamber pressure and temperature changes and thrust generated by the pulsejet.

The MATLAB S-function code could be found in Appendix A.
5.3 MATLAB model

A discrete model was generated to simulate the pulsejet process. There are 3 layers in the Matlab model.

As shown in Figure 5-1, the first layer is the interface layer change information between human and the computer. Through this layer, one could set the parameters of the pulsejet process, such as environment pressure, temperature, free stream speed and geometry of the pulsejets. Also through this layer, one could decide the input fuel flow rate and get the results interested from the simulation. In this program, if set the fuel flow rate more than needed, a perfect fuel flow rate could be calculated, based on the fresh air entered into the combustion chamber. In this work, combustion chamber pressure and temperature, thrust, fuel consumption and operational frequency were chosen as results to indicate the performance of the pulsejet.

\[ \text{Figure 5-1: First layer of the Matlab simulation} \]

The second layer is the major layer to complete the simulation process. Discrete simulation model was used in this layer. As shown in Figure 5-2, a valveless pulsejet was divided into 3 parts, the combustion chamber, inlet tube and exhaust tube.
Figure 5-2: The second layer of the Matlab simulation

The inlet tube and exhaust tube are added into the model as S-function Module. The Module use 1-D CFD codes to calculate the flow filed in the tubes. They take the combustion chamber pressure and temperature from the combustion chamber part and use them as boundary conditions. For simplicity Van Leer type scheme was used in the 1-D CFD code. The results of the calculation were used to calculate the thrust generated by the pulsejet as well as the velocity, temperature and oxygen mass fraction of the flow into the combustion chamber. The fresh air is supposed as enter the combustion chamber only from the inlet tube. It is calculated by the help of the integrate module ‘integrator 2’. The initial condition of the integrate module is set to be zero; as the hot gas from the combustion chamber enter the inlet
tube from the combustion chamber, the integrate value increase until the hot gas totally occupy the volume of the inlet tube; as the fresh air enter the inlet tube from the inlet, the integrate value decrease until it reaches zero, which indicate that the inlet tube is fully loaded with fresh air; the fresh air enters into the combustion chamber after the inlet tube is fully loaded.

The combustion chamber, however, is totally constructed by the simulink blocks. If consider the combustion chamber part as a black box, the inputs into the box are initial heat addition, fuel flow rate and flow from inlet and exhaust tubes. By using equations (5-4) and (5-9), the combustion chamber pressure adjustment was calculated in each step and added to the previous pressure value to get the current chamber pressure. Temperature is calculated in the same manner as pressure value by using equations (5-5) and (5-10). Temperature value also considered the lower temperature flow into the combustion chamber. The mass ratio of the flow into the combustion chamber in a single step and the fluid already in the combustion chamber is used to evaluate the temperature change brought from the cold flow into the combustion chamber.

The third layer simulates the combustion in the combustion chamber, shown in Figure 5-3. The input of this layer is fuel flow rate into the combustion chamber, fresh air enter the combustion chamber and combustion temperature, output is the resulted heat addition to the combustion chamber. In this work, the combustion process is greatly simplified. One step combustion model was used, heat value and AF ratio of the fuel were added as parameters of
the combustion layer. The chemical reaction is believed much faster than the transport process in the combustion considered. The reaction rate was thus dominated by the mass transport process, and was set to be related to the contact area between reactants, turbulence strength and combustion chamber temperature.

5.4 Results and discussion

5.4.1 Validation of Matlab simulation

By disconnect the tailpipe, the Matlab model could also model the Helmholtz pulsejet. Only one pipe included in the simulation, the Helmholtz pulsejet is simpler to model than the
valveless pulsejet. To compare with CFX simulation results, 50 cm Helmholtz pulsejet with 0.0123 m$^3$ chamber volume, 0.25 m tailpipe length and 0.023 m tailpipe diameter was simulated. Figure 5-4 shows the combustion chamber pressure together with the CFX results, mentioned in CHAPTER 4. From the picture, the Matlab simulation provided the same frequency and pressure level with that of the CFX simulation.

![Figure 5-4: Chamber pressure of the 50 cm Helmholtz pulsejet](image1)

![Figure 5-5: Chamber pressure of the 50 cm pulsejet with 10cm long and 1.6cm diameter inlet](image2)
Comparison also carried out for valveless pulsejet. Figure 5-5 provided the MATLAB results for combustion chamber pressure for valveless pulsejet with 10 cm long and 1.6 cm diameter inlet compared with CFX and experimental results, as results in CHAPTER 2. The Matlab simulation also provided similar pressure magnitude and frequency.

Although there is still difference between the results of the Matlab simulation and the CFX simulation, the similarity of these results provided that the model is quite accurate in predicting the frequency of the pulsejet. Because the pulsejet process is based on the acoustic phenomenon, which appears as operational frequency, the principle that uses the Matlab simulation to get quick results is validated.

5.4.2 Matlab simulation on valveless pulsejet with different environmental conditions

As a propulsion device, pulsejet could be running in different environmental conditions. The altitude and speed pulsejet running could have great effects on its performance. In the Matlab model, environment temperature, pressure and free stream speed could be set in the parameter.

The pressure and temperature could be used directly in the calculations. The free stream speed, however, is achieved by setting the boundary condition of the inlet tube as stagnation pressure and stagnation temperature of the free stream flow.
Figure 5-6 shows the simulation results of BMS Hobby scale pulsejet with a 15 cm long, 1.5 cm diameter inlet running in different free stream velocities. Figure 5-6a shows the operational frequency and thrust generated by the pulsejet. As the free stream velocity increased from 0 to 140 m/s, the negative thrust generated by the flow through the inlet kept the same level and then decreased, the positive thrust generated by the flow from the exit increased to its peak value of 3.1 N at 120 m/s and then began to drop. The total thrust
increase as the forward speed increasing. The operation frequency of the pulsejet obeyed the equation developed in CHAPTER 2, increased with the free stream velocity.

In Figure 5-6b, the Fuel consumption and ISP are displayed. As forward speed increasing, the fresh air enters the combustion chamber increased and led to fuel consumption increasing. The ISP reached a peak value of 553 second around 90 m/s and decrease after that.

Further increase the free stream velocity caused the pulsejet failed, the pulsation combustion feature lost. As results, pulsejet fuel consumption increased to 1.3 g/s, thrust decreased to 1.5 N and ISP decreased to 120 s for 150 m/s case compared to 0.51 g/s, 2.3 N and 461 s for 140 m/s case. The results clearly illustrated that the pulsed combustion propulsion provides much better performance than steady state propulsion.

Simulations also carried out for pulsejet running in different altitude with standard pressures and Temperatures. As shown in Figure 5-7, the forward speed has much larger effects on the pulsejet performance than the altitude does. When the forward speed exceeds 140 m/s, the pulsejet failed to work as an unsteady propulsion system. It makes the fuel consumption increased and thrust decreased sharply and thus ISP decreased to a low lever around 160 s. The performance for the valveless pulsejet with this particular inlet gets its best value at around 90 m/s for sea level pressure and temperature. For higher altitude of more than 500 m above sea level, there are two speed regions providing best performance, speed around 90 m/s and 130 m/s.
5.4.3 Matlab simulation on valveless pulsejet with different inlet geometries

The inlet design is very important for a valveless pulsejet. From the conclusion drawn in previous chapters, frequency and volume ratio is important for the inlet of a valveless pulsejet. Simulations were carried out by CFX model in previous chapters to discuss the relationship between the inlet geometry and the valveless pulsejet performance. However, the CFX simulation is time costing. Matlab simulation is not precision enough, but could capture the trend of how the inlet geometry effect on the pulsejet performance.

Figure 5-8 shows the thrust and frequency of the pulsejet for different inlet lengths and diameters. From Figure 5-8a and Figure 5-8b, the inlet thrust and outlet thrust decreased sharply when diameter at around 0.023 m. This decreasing is caused by the sharply pressure level decreasing at this diameter. For the same geometry simulated in CFX, the running of the pulsejet died out. Compare the outlet thrust with the Frequency, could find that the largest thrust happened when the frequency is around 220 Hz, this is compatible with the exhaust
frequency for this combustion chamber and tailpipe. Another factor effects the outlet thrust is the inlet and combustion chamber volume ratio. For this 50 cm pulsejet, the optimal volume ratio is around 0.128, matching with the value provided in CHAPTER 3.

Figure 5-8: effects of inlet geometry on the Total Thrust of valveless pulsejet
(a) negative thrust from inlet; (b) thrust from outlet; (c) Total thrust; (d) Frequency

Unit: length in m, thrust in N, Frequency in Hz

Figure 5-8c shows the Total thrust, which is calculated by the exhaust thrust minus the inlet thrust. According to the picture, the largest total thrust happened at a suitable inlet diameter
to allow enough fresh air enter the combustion chamber and prevent large inlet thrust with frequency and volume ratio close to the optimal value.

5.4.4 Matlab simulation on inlet lengths and velocities

As mentioned in CHAPTER 2, frequency match is important in valveless pulsejet performance. When it comes to inlet frequency, it is influenced by two factors, geometry and free stream velocity. In practical, a valveless pulsejet with changing inlet length is possible for different free stream velocity to keep the inlet frequency match the exhaust frequency. So the pulsejet running in different free stream velocity with different inlet length is interesting.

Figure 5-9: Performance of valveless pulsejet with different inlet length and free stream velocities
(a) Total thrust; (b) Operational frequency.

Figure 5-9 shows the simulation results of BMS Hobby scale pulsejet with 1.5 cm diameter inlet and different inlet length running in different free stream velocities. Figure 5-9(a) is total thrust generated. The polyline in the figure is generated by connecting peak thrust value
at different free stream velocity. The inlet length increased with free stream velocity to keep
the valveless pulsejet optimized. Figure 5-9(b) shows the operational frequencies.

Recall

\[ f_i = \left[ c_i / (2\pi) \right] [S_i / (VL_i)]^{0.5} (1 - M)^{-1.5} \]  

(2-4)

Frequency in the figure is increasing with free stream velocity and decreasing with increasing
inlet length, as expected. The polyline is directly copied from Figure 5-9(a). It is very
interested that the polyline is identical to the iso-frequency line, which supporting the
frequency match theory in CHAPTER 2 again.

Need to mention that frequency here is not proportional to \((1/L_i)^{0.5} (1 - M)^{-1.5}\). The average
sound speed \(c_i\) in the inlet tube is changing with Mach number and inlet length. As Mach
number increasing, more fresh air will occupy the inlet tube for a longer period, and thus
average sound speed decreased. In the range studied in Figure 5-9, frequency could be
approximately described as proportional to \((1/L_i)^{0.5} (1 - M)^{-0.5}\).

5.5 Conclusions

From the results of both Matlab and CFX simulations, the following conclusions are
obtained:

1. The Matlab simulation based on 1-D flow in the pipes and simplified models in
   combustion chamber works well in capture frequency and pressure of the pulsejet
   running process.
2. Performance of valveless pulsejet increase with free stream velocity in low speed region and failed to keep unsteady propulsion above some particular velocity. The pulsed propulsion provides much better performance than steady state propulsion.

3. Altitude has much less effect on the valveless pulsejet performance than the free stream speed. At sea level, valveless pulsejet running best in 90 m/s free stream velocity; at altitude of more than 500 m above sea level, valveless pulsejet running best in 90 m/s and 130 m/s free stream.

4. Both inlet chamber volume ratio and inlet frequency have effect on the valveless pulsejet performance. The largest outlet thrust happened when the inlet frequency match the outlet frequency and the inlet chamber volume ratio at around 0.128. The largest total thrust happened at a smaller inlet diameter to avoid large inlet thrust.

5. While free stream velocity increasing, inlet length should be increased to help the inlet frequency match the exhaust frequency to maintain peak performance. The peak performance frequency keeps the same for different velocities, as expected.
REFERENCES


CHAPTER 6. NUMERICAL STUDY OF PULSEJET OPERATION IN AN ENCLOSURE

6.1 Introduction

The pulsejet is an unsteady propulsion device that generates intermittent thrust [1]. Unlike the traditional jet engine, the pulsejet is an acoustically resonant device and the acoustic wave makes four trips during a cycle. Heat release corresponding with the compression acoustic wave in the combustion chamber, heat addition occurs isochorically rather than isobarically during the combustion process. For this reason, the pulsejet is used to simulate constant volume combustion. This is currently of interest because constant volume offers an increase in thermodynamic efficiency when compared with isobaric combustion. In this paper, we discuss a pulsejet operation in an enclosure to mimic an unsteady combustor that may be used in a gas turbine, replacing the steady, constant pressure combustor now in use.

The pulsejet concept was first developed in the beginning of 20th century. In 1930’s, the first working pulsejet was designed and used to power German V-1 weapon in World War II. A lot of research was thus carried out on pulsejet in the following decade. However, the efficiency of pulsejet was relatively low compared to turbojet and it was put aside for decades.

However, interest in pulsejets has increased recently because they exhibit many of the features found in more complex unsteady flowfields. The major advantage of the pulsejet is its simplicity; the only moving parts of the pulsejet are the reed valves. The reed valve is
operated passively by the combustion chamber pressure; when the pressure is lower than atmospheric, the valve is opened, when the chamber pressure is higher than atmospheric, the valve is closed to prevent hot gas from leaving the combustion chamber from the inlet. Without complicated compression system, pulsejet can achieve high thrust to weight ratio, which makes it an attractive candidate for a micro propulsion system.

In [2-4], a combined numerical and experimental approach was used to obtain detailed information of the unsteady processes occurring during pulsejet operation. Due to the inertia and elasticity of reed valves, there is a time delay between the combustion chamber pressure decrease below atmospheric pressure and the moment when the valves are totally opened. For the pulsejet running at a frequency over 200 Hz, this delay becomes important. In ref. [5], the dynamics of the reed valve operation was included in the numerical model, which significantly improved predictability of the model with respect to pulsejet performance.

There is also interest in using pulsed combustion devices, such as pulsejet combined with an ejector, for achieving pressure-gain heat addition in gas turbine system [6-9]. In a traditional gas turbine, isobaric heat addition to the combustion chamber actually suffers a loss in total pressure across the combustion chamber. By using a pulsejet in the combustion chamber to achieve isochoric heat addition, Paxson and Dougherty [9] obtained a slight increase in total pressure in the combustion chamber, which then effectively increases the isentropic efficiency of the compressor, increasing the overall efficiency of the engine.
This paper develops a numerical model to simulate the pulsejet running in an enclosure. Different geometries and boundary conditions are simulated to analyze how the outside conditions effect the pulsejet performance. This is used to explain the pressure rise observed by Paxson & Dougherty.

6.2 Numerical model

In this paper, the commercially available CFD package CFX is used to simulate a pulsejet running in an enclosure. To simulate the pulsejet running in a shroud, not only the flow field inside the pulsejet, but also the flow field outside the pulsejet needed to be simulated. Similar as in CHAPTER 2, a two-dimensional model is developed to simulate the flow inside and outside the pulsejet.

As shown in Figure 6-1, due to the axis-symmetrical geometry of the flow field, only a 4 degree wedge was simulated to save the computational time. The boundary condition at the
inlet of the shroud was set to be constant pressure and temperature, which determines the pressure and temperature around the running pulsejet. This allowed for the simulation of air coming from a compressor at elevated temperature and pressure. The boundary condition at the outlet of the shroud was set to be constant pressure or mass flow rate, controlling the bypass ratio around the pulsejet.

Resistant layers were added to inlet and outlet of the shroud so that acoustic reflections from the inlet and outlet are minimized and allowing simulation to reach quasi-steady performance more quickly [10]. As shown in Figure 6-1, the boundary pressure $P_{\text{set}}$ is set at the outside of the resistant layers. The averaged pressure $P_{\text{in}}$ and $P_{\text{out}}$ at inside of the layers are different from $P_{\text{set}}$, due to the existence of the resistant layer. The difference between $P_{\text{out}}$ and $P_{\text{in}}$ represents the pressure rise from the inlet to the outlet of the shroud.

A smaller mesh size was generated for the flow field inside the pulsejet and the region near the ejector to acquire better results, compared with the external flow field. The computations were performed on the NCSU IBM Blade Center utilizing a single 3.0 GHz Inter Xeon processor. Typical computational time for one cycle of the pulsejet was about 30 CPU hours, and multiple cycles were simulated to assure a quasi-steady solution had been achieved.

The flow field in the intake head of the pulsejet was not simulated; rather, the flow into the combustion chamber through the intake head was computed by a FORTRAN subroutine. This was done so that simulate at the reed valves and the fluid-solid interaction was not required. The mass flow from the inlet was a result of the following coupled equations [5]:
\[
\frac{\dot{m}}{\dot{m}} = \frac{A_m}{L_m}(P_0 - P_{up}) \quad (6-1)
\]
\[
\frac{dV_v}{dt} = \left[ A_m (P_{up} - P_{cc}) - k_v x_v \right] m_v \quad (6-2)
\]
\[
\frac{dx_v}{dt} = V_v \quad (6-3)
\]
\[
P_{up} = P_{cc} + \frac{m^2}{2 \rho} \left( \frac{1}{A_v} - \frac{1}{A_m} \right) \dot{m} \quad m > 0.0 \quad (6-4)
\]
\[
P_{up} = P_{cc} + \frac{\dot{m}^2}{2 \rho A_v}, \quad m < 0.0 \quad (6-5)
\]
\[
A_v = \max(A_{v_{min}}, \alpha x_v) \quad (6-6)
\]

where \( P_0, P_{up}, \) and \( P_{cc} \) are pressure at the inlet of the head, at the upstream side of the valve and at the combustion chamber, respectively; \( \dot{m} \) is the desired mass flow rate into the combustion chamber; \( x_v, V_v, \) and \( m_v \) are position, velocity and mass of the valve, respectively; \( A_v \) is the open area of the valve decided by the valve position.

At the beginning of every time step of the simulation, the mass flow rate was calculated. Due to the complexity of the coupled equations, a two step approximate method was used. The first step used equations (6-2) and (6-3) to figure out the position of the valve, and used equation (6-6) to get the open area of the inlet. The second step used iteration method to solve equations (6-1), (6-4) and (6-5), get a mass flow rate.
In the complementary experiments, fuel is injected and mixed with air in the head of the pulsejet, entering the combustion chamber as premixed fuel air mixture. In this paper, the mixing process is not simulated. The stoichiometric propane air mixture is injected into the combustion chamber with a mass flow rate calculated by equations (6-1) to (6-6). When the reactants mix with the residual hot products from the previous cycle in the combustion chamber, the temperature of the mixture increases due to heat and mass transfer to above the auto-ignition temperature and the reactants are consumed. Since the combustion process is believed to be controlled by the heat and mass transfer process, the eddy dissipation model was chosen as the combustion model.

Both gas phase and multi-phase combustion process were simulated. Propane was chosen as fuel for gas phase reaction in some simulations, liquid JetA was chosen for multi-phase combustion in other simulations. NOx generation was also considered in multi-phase combustion process, both prompt NO formation scheme and thermal NO formation scheme were considered.

6.3 Results and discussion

6.3.1 Gas phase combustion by using Propane as fuel

A pulsejet running in a shroud, without an ejector, was simulated first. The combustion chamber pressure of the pulsejet was compared with the case of a pulsejet running in the open air, as shown in Figure 6-2. This pressure-time profile shows that the frequency for the case of a pulsejet running in a 15 cm diameter shroud is higher; also, the peak pressure is
lower than that for the pulsejet in open air. The frequency for the former case is 259 Hz compared with 213 Hz for the later one. The observed 22% increase in frequency can be explained by looking at the temperature difference in the area around the exit of the pulsejet. The pulsejet is driven by the acoustic phenomena, with the acoustic wave traveling back and forth in the pulsejet helping to establish the pressure oscillations in the combustion chamber, and thus achieving the unsteady combustion process. When the combustion chamber pressure decreases to below atmospheric, not only does fresh reactants enter the combustion chamber through the reed valves, but also ambient air enters the tailpipe from the exit of the pulsejet. The acoustic wave speed, which is a function of the temperature, is decreased by travelling through the slug of fluid which contains ambient air at low temperature. For the pulsejet running in a shroud, however, hot exhaust gases accumulate around the exit of the pulsejet, as shown in Figure 6-3a. So, during the low pressure portion of the cycle, higher temperature exhaust gas was pulled into the pulsejet, which leads to higher acoustic wave speed and therefore higher operating frequency. In the open air, as shown in Figure 6-3b, due to fresh air supplied radially around the pulsejet, the high temperature exhaust does not accumulate around the tailpipe exit.

This higher frequency leads to a smaller amount of reaction mixture entering the combustion chamber every cycle and thus a lower average chamber pressure, resulting in a reduction in thrust by about 20%, from 21.3 N to 17 N.
A pulsejet with an ejector was also simulated. As explained in [6], because of the transient nature, a starting vortex is generated from the exit of the pulsejet. This vortex moves...
downstream and impacts the nose of the ejector, creating a low pressure region in front of the nose, and generating additional thrust. These ejectors, which only work for unsteady flows, can increase net thrust by more than 50%. Since one of the objectives is to increase pressure, any increase in net thrust will manifest itself as an increase in pressure at the exit plane. There are several geometrical parameters for the ejector, as shown in Figure 6-4. Following the results from [7], in this paper, the parameters were chosen as follows: \( \delta/D_I = 2 \), \( D_{ei}/D_I = 2.4 \), \( r/D_I = 0.6 \), and \( L/D_I = 8.6 \).

![Figure 6-4: Geometry of the ejector](image)

Figure 6-5 compares the temperature and pressure of the flow field around the exit of the pulsejet, with an ejector and with or without an enclosure. The upper half depicts the temperature field, changing between 300 K and 1000 K; the lower half depicts the pressure field, changing between \( 8 \times 10^4 \) Pa and \( 10^5 \) Pa.

As shown in Figure 6-5a, by using the ejector, the hot gas from the pulsejet was effectively directed to the exit of the shroud, the temperature around the exit of the pulsejet is still higher than in the open air (Figure 6-5b), but much lower than for the case running in the shroud.
without an ejector. The frequency was calculated to be 233 Hz, which is a 9.4% increase compared to a pulsejet running in the open air.

Figure 6-5: Temperature and pressure around the exit of the pulsejet with ejector.
(a) in shroud; (b) in open air

The shroud pressure and temperature can be set by changing the inlet boundary conditions of the shroud. Table 6-1 summarizes results for a pulsejet running in different shrouds and with different boundary conditions. The Pressure ratio is non-dimensional pressure promotion through the shroud, defined as $\frac{\text{Average}(P_{\text{out}} - P_{\text{in}}) \cdot A}{\text{Total Thrust}}$, where $A$ is the cross sectional area of the shroud. Total thrust is the sum of the thrust on pulsejet and the thrust on ejector.
Table 6-1: Results for a pulsejet running in shroud with different geometries

<table>
<thead>
<tr>
<th>Shroud pressure (10^5 Pa)</th>
<th>Shroud diameter (cm)</th>
<th>Ejector</th>
<th>Frequency (Hz)</th>
<th>Averaged (Pout – Pin) (Pa)</th>
<th>Bypass ratio</th>
<th>Thrust on Pulsejet (N)</th>
<th>Thrust Augment</th>
<th>Pressure ratio a</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open air</td>
<td></td>
<td>No</td>
<td>213</td>
<td>0</td>
<td>N/A</td>
<td>21.31</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yes</td>
<td>219</td>
<td>0</td>
<td>19.09^b</td>
<td>21.31</td>
<td>1.96</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>15.2</td>
<td>No</td>
<td>259</td>
<td>763</td>
<td>4.35</td>
<td>17.01</td>
<td>1</td>
<td>0.68</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yes</td>
<td>233</td>
<td>2967</td>
<td>7.26</td>
<td>24.13</td>
<td>2.55</td>
<td>0.82</td>
</tr>
<tr>
<td></td>
<td>11.4</td>
<td>Yes</td>
<td>241</td>
<td>4033</td>
<td>4</td>
<td>20.46</td>
<td>2.0</td>
<td>0.94</td>
</tr>
<tr>
<td>1.5</td>
<td>15.2</td>
<td>Yes</td>
<td>234</td>
<td>4585</td>
<td>9</td>
<td>35.57</td>
<td>2.88</td>
<td>0.77</td>
</tr>
</tbody>
</table>

^a equals to \( \frac{\text{Avg}(P_{out} - P_{in}) \times A}{\text{Total Thrust}} \)

^b calculated by \( \frac{\text{flow through ejector}}{\text{flow through pulsejet-1}} \)

For a 10^5 Pa shroud with pressure exit boundary, because of a similar condition at the inlet of the pulsejet, the mass flow through the pulsejet into the shroud is the same as in the open air. The bypass ratio, however, decreased from 19.09 to 7.26. This can be explained by the fact that for the case running in shroud, there is no ambient air available to be entrained into the ejector. This also causes the pressure at the nose of the ejector to be lower than for the case in the open air, thus generating larger thrust on the ejector. As shown in Figure 6-5a, low pressure region formed in the front of the ejector; in Figure 6-5b, however, air coming from around makes the pressure in the front of the ejector keeping atmospheric. Based on the results shown in Table 6-1, the presence of the ejector for a pulsejet running in a shroud provides nearly a 300% increase in pressure from the inlet to the outlet of the shroud. These results are in a very good agreement with experimental results reported in ref. [9]. Decreasing
the mass flow rate at the exit can further increase the pressure. However, this pressure increase cannot exceed a certain limit, which determined by the total thrust on the pulsejet and the ejector. If the shroud is considered as a control volume, the pressure difference at the inlet and outlet will generate a force on the control volume. This force together with the thrust generated by increasing speed of the flow through the shroud should be equal to the sum of forces on all of the devices in the shroud, including the pulsejet, ejector and the shroud itself. Because the force on the shroud and the outer side of the pulsejet is a drag force, the pressure force on the volume must be smaller than the total force on the pulsejet and the ejector.

By the analysis made above, decreasing the cross sectional area of the shroud is a potential method to increase the pressure rise across the shroud for a given thrust level. However, from Table 6-1, with the pulsejet running in an 11.4 cm diameter shroud, the thrust on the pulsejet and thrust augment by ejector also decreases, and thus the pressure rise is limited.

Table 6-2: Results for pulsejet running in 15 cm diameter, with different boundary conditions

<table>
<thead>
<tr>
<th>Shroud exit boundary condition</th>
<th>Frequency (Hz)</th>
<th>Averaged (P_{out} - P_{in}) (Pa)</th>
<th>Bypass ratio</th>
<th>Thrust on Pulsejet (N)</th>
<th>Thrust Augment</th>
<th>Pressure ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure</td>
<td>233</td>
<td>2967</td>
<td>7.26</td>
<td>24.13</td>
<td>2.55</td>
<td>0.82</td>
</tr>
<tr>
<td>0.18 kg/s</td>
<td>232</td>
<td>2996</td>
<td>6.5</td>
<td>23.38</td>
<td>2.65</td>
<td>0.83</td>
</tr>
<tr>
<td>0.09 kg/s</td>
<td>230</td>
<td>3543</td>
<td>2.9</td>
<td>21</td>
<td>2.93</td>
<td>0.98</td>
</tr>
</tbody>
</table>
For the pulsejet running in the same shroud, a different bypass ratio could also provide different pressure rise across the shroud. As shown in Table 6-2, lower bypass ratio causes a lower thrust on pulsejet, but higher thrust augmentation on the ejector, which provided similar total thrust on the combination of pulsejet and ejector. Due to the drag force on the pulsejet and shroud was decreased, the pressure rise and thus the pressure ratio increased with the decreased bypass ratio.

![Figure 6-6: Pressure ratio vs. Bypass ratio](image)

This conclusion was also valid for all the results of the pulsejet with an ejector, the larger the bypass ratio (defined as mass through shroud but around pulsejet divided by mass which goes through the pulsejet), the larger the drag force on the outer side of the pulsejet and the shroud, and thus the smaller the pressure ratio (Figure 6-6).
6.3.2 Multi-phase combustion and NO formation

Multiphase combustion process with Liquid JetA was also simulated. Figure 6-7 is steady state result for liquid jetA burning in pulsejet, the upper part is the gas phase JetA mass fraction and the lower part is the oxygen mass fraction.

![JetA Mass Fraction (Plane 1)](image1)

![O2 Mass Fraction (Plane 2)](image2)

Figure 6-7. Steady state result for liquid JetA burning in pulsejet

Liquid JetA of 10 μm diameter was injected into the combustion chamber together with the fresh air. Different from gas phase JetA input, there is a time delay for the vaporization of the liquid JetA, which lead to the detachment of the jetA from the Oxidizer, as shown in the picture. The combustion process is thus doing like a diffusion combustion process rather than a premixed combustion.

Figure 6-8 is transit simulations for liquid jetA combustion in pulsejet. Figure 6-8a is JetA of 10 μm diameter injected with fresh air and Figure 6-8b is JetA of 20 μm diameter. For both cases, jetA was detached from Oxygen, as shown in the pictures. However, for the 10 μm case, jetA vaporized faster, it is used up before they leave the pulsejet. The 20 μm case is different, parts of jetA leaved the pulsejet before consumed. For this case, the unconsumed jetA is 12.4% of the total jetA injected into the combustion chamber.
Figure 6-8. Pulsejet running with liquid JetA of different diameters as fuel.
*(a)* 10 μm; *(b)* 20 μm.

Due to this partially consumed combustion process in the pulsejet, performance of pulsejet running with 20 μm diameter JetA dropped sharply from that of the pulsejet running with 10 μm diameter JetA. Figure 6-9 is the thrust and combustion chamber pressure of pulsejet change with liquid JetA diameter. Pressure is gauge pressure normalized by the atmospheric value, thrust is normalized by the thrust of pulsejet running with gas phase JetA.
With 5 μm liquid JetA, pulsejet generated a larger thrust than gas phase JetA. This is due to larger density for mixture includes liquid JetA than gas phase JetA, which lead to higher mass flow rate of reactants into the combustion chamber. Thrust then decrease with the diameter of liquid JetA. When the diameter of JetA increased to 50 μm, pulsejet could no longer operate.

Model for NO generation is also included in the simulations. NO mass fraction is calculated to be around $1.5 \times 10^{-5}$ for both steady state and transit simulations. Decreasing of NO formation by pulsed combustion is not observed. Combustion model in CFX is considered too simple for this particular task.

### 6.4 Conclusions

From the analysis presented above, the following conclusions are obtained:

1. Hot exhaust gases can accumulate around the exit of the pulsejet running in a shroud, which causes an increase in pulsejet operating frequency by about 20% compared to the pulsejet running in open air. This frequency increase decreases the thrust generated by the pulsejet by about 20%.

2. The pulsejet with an ejector running in the shroud can effectively direct the hot exhaust gases through the ejector, preventing them from accumulating around the exit of pulsejet, lowering the frequency of the pulsejet with the ejector. The frequency is lower than without the ejector but still higher than in the open air. A configuration
with an ejector in a shroud generates larger thrust, compared to configuration with an ejector in the open air.

3. A pulsejet operating in open air does not produce a pressure rise. By enclosing the pulsejet in a shroud, a net pressure increase is possible. This pressure rise can be greatly increased by adding an ejector to the pulsejet. In all cases, the pressure rise will scale with (but always less than) the net thrust.

4. For the pulsejet in a shroud with the same ejector, lower bypass ratio leads to higher pressure rise between inlet and outlet of the shroud. However, this pressure rise could not exceed the limit determined by the thrust generated by the pulsejet and ejector and the cross sectional area of the shroud.

5. Liquid fuel makes combustion more difficult. Pulsejet performance is sensitive to the diameter of inject fuel.
REFERENCES


CHAPTER 7. NUMERICAL STUDY OF A PULSEJET-DRIVEN EJECTOR

Nomenclature

\( \alpha \)  \quad \text{expansion angle of the ejector}

\( d \)  \quad \text{pulsejet tailpipe diameter}

\( D \)  \quad \text{diameter of ejector at the smallest position}

\( \delta \)  \quad \text{distance between the ejector nose and the exit of the pulsejet}

\( \varphi \)  \quad \text{thrust augmentation}

\( L \)  \quad \text{length of the ejector}

\( r \)  \quad \text{radius of the nose of the ejector}

\( x \)  \quad \text{distance between the ejector throat and the exit of pulsejet (equals to } \delta + r \text{)}

7.1 Introduction

Ejectors are passive devices that are used to increase the thrust of propulsion systems through entrainment and fluidic work exchange. When the propulsion system is pulsed (i.e. unsteady), the augmentation of thrust can be substantially larger than if it is steady, although the mechanism for this enhanced performance is still not completely understood. For reference, thrust augmentation, \( \varphi \), is defined as the thrust of the combined system (the jet plus ejector) divided by that of the driving jet alone. Examples of pulsed propulsion systems
include pulsejets and pulse detonation engines (PDE). Much of the early experimental work on pulsejets and pulsejet-ejector combinations was carried out by R. Lockwood in the 1960’s[1]. In recent years, interest in pulsed ejectors has been renewed as interest in pulsejet and PDE technologies has increased.

Several experimental studies were carried out to quantify the performance of ejectors for PDEs [2-7]. In Ref. [8], an analytical model was developed which showed that the pulsed ejector augmentation was closely related to the vortex generated by the pulsed source. This relationship was verified by the experimental particle imaging (PIV) technique performed on several pulsed thrust systems, and described in Ref. [9].

In the present work, a computational model was developed to study the physics of unsteady thrust augmentation. The model simulates a particular pulsejet-driven ejector system on which extensive entrainment, augmentation, and flowfield experiments have been performed[10-12]. A pulsejet is a good pulsed propulsion source for investigating ejector performance. It is relatively easy to construct and operate. More importantly, it provides a high enthalpy, highly impulse exit flow, which it can yield exceptionally high augmentation levels.

7.2 Numerical Model

A commercially available CFD package, CFX 11, was used to model the combustion and flow inside the valved pulsejet, and the flow field downstream of the pulsejet, to gain more understanding of how the pulsed flow interacts with the ejector. This package was used
successfully in the work of Ref. [13], where the internal and downstream flowfields of the pulsejet were simulated without an ejector present. In that study, the simulated pulsejet thrust, operating frequency, combustion chamber pressures, and emitted vortex characteristics were found to match experimental measurements very well.

The dimensions of the modeled system were the same as in the experiment described in Ref. [10]. A small pulsejet (50cm in length) was used as the pulsed source. The tailpipe diameter and the exit diameter of the pulsejet were 3.18 cm and 3.68 cm, respectively.

A two-dimensional axi-symmetric model was utilized to achieve an affordable simulation time. The inlet boundary condition for the pulsejet was a temporally varying mass flow rate determined by a coupled, lumped volume inlet and a reed valve model [13], as explained in CHAPTER 6.

![Figure 7-1. Mesh generation for combined pulsejet and ejector simulations](image)

The mesh used for the computation was shown in Figure 7-1. Refinement (i.e. grid resolution) was provided in all regions where boundary layers, shear layers, or large pressure
gradients dominate the flow. These include regions near the wall and at the front of the ejector. The $k-\omega$ turbulence model was used for simulating turbulent flow field [14]. The thrust on the ejector was calculated by integrating the pressure and shear forces along the surface of the ejector.

A premixed, stoichiometric propane/air mixture was assumed for the pulsejet operation. A 5 step reaction scheme provided by CFX was adopted, including propane oxidation, CO oxidation, H$_2$ oxidation and both directions of the water-gas reaction. The chemical reaction timescale is believed to be much smaller than the timescale for the heat and mass transfer for the pulsejet simulated in this work; the eddy-dissipation combustion model was used to simulate the pulsejet combustion process [15].

Computations were performed on the NCSU IBM Blade Center utilizing a single 3.0 GHz Inter Xeon processor. Typical computational time for one cycle of the pulsejet was about 30 CPU hours, and multiple cycles were simulated to assure that a quasi-steady solution was achieved.

7.3 Results and Discussion

As shown in Figure 7-2, there are several parameters characterizing the geometry of the ejector. For the present study the following geometry was used as the base case geometry: $\delta/d = 2$, $D/d = 2.4$, $L/d = 8.6$, $r/d = 0.6$, and a 4.0 degree expansion angle in the duct. Considering the interaction of ejector with the vortex generated by the pulsejet, another parameter, $x/d$, as shown in Figure 7-2, was also used in this study. $x/d$ is directly related to
\( \delta/d \), but compared to \( \delta/d \), this value provides the relationship between vortex and the ejector more straightforward.

![Figure 7-2. Dimensions of the ejector](image)

The flow field downstream of the pulsejet exit was investigated, especially the vorticity and pressure fields. It was observed that a vortex ring was generated at the pulsejet exit and forms a low pressure region at the ring. At the same time, a high pressure region was produced downstream of the vortex ring, as shown in Figure 7-3 (It should be noted that the high pressure region is part of the vortex. It was also observed to be generated and faded together with the vortex downstream of the running pulsejet without an ejector.). This vortex ring

![Figure 7-3. Pressure distribution generated by the vortex ring](image)
moved toward the ejector. If the ejector diameter was chosen correctly, the high pressure region was pushed into the ejector and the low pressure region was attached to the front of the ejector. This low pressure region generates thrust. In our simulations, this low pressure could be 25% lower than the atmospheric pressure, generating 130 N peak thrust while the vortex attaching to the nose of the ejector.

As shown in Figure 7-4, moving average of thrusts and combustion chamber pressure over a cycle was observed and used to decide the stabilization of the numerical solutions. Several cycles were needed to reach a quasi-steady solution. For the pulsejet with the base case ejector, as shown in Figure 7-4, total thrust is stabilized at 42 N, thrust on the pulsejet was 21 N and thrust augmentation was calculated to be 2.0. The thrust on the pulsejet exhibits a slight variation (±3%) for different cases. This variation was ignored in this study, 21 N was used as the thrust on the pulsejet in all cases to calculate the thrust augmentation.

Simulations of a straight ejector (with a constant cross-sectional area) were performed to compare with the base case ejector that has a 4 degree expansion duct. The thrust augmentation was calculated to be 1.72 for straight ejector, 4.5% smaller than the experimental result of 1.8. Results indicated that for the base case ejector, a 4 degree expansion in the duct provided a 16% increase in thrust augmentation compared with the straight ejector. This trend was comparable with an 11% increase observed in experiments.

It was observed that the majority of the augmentation was actually generated during the entrainment period, which occurred after the emitted vortex had passed through the ejector,
decayed, and accelerated the fluid which was in front of it. The ejector then refilled, creating suction on the rounded inlet (thrust).

![Graph of Chamber Pressure vs Time]

*Figure 7-4. Moving average pressure and thrusts for pulsejet with the base case ejector*

Figure 7-5 shows the thrust on the ejector during a cycle for the base case ejector. At point 1 the high pressure region reached the nose of the ejector and negative thrust was generated; at point 2 the vortex attached to the nose of the ejector and large positive thrust was generated;
point 3 is the moment of time when the acoustic wave went through the ejector, the vortex was initiated at the exit of the ejector and the expansion wave was reflected back; at point 4 hot gas together with fresh air that were sucked into the ejector finally passed through the ejector. The time from point 1 to point 4 represents 60% of the one cycle, 80% of the thrust on the ejector was generated during this period. This computationally observed behavior was consistent with experimental results reported by Heffer et. al.[16]

![Thrust on the ejector during one single cycle](image)

*Figure 7-5. Thrust on the ejector during one single cycle*

### 7.3.1 Diameter Optimization

Since the vortex was so important for the performance of the ejector, the position of the vortex ring (determined by the core of the vortex, approximately positioned at the point where velocity is zero) was observed and marked out for the situation in which the pulsejet was running without the ejector. As shown in Figure 7-5, the red line represents the diameter of the vortex ring (twice the distance from the core of the vortex to the centerline) as a
function of distance from the exit of the pulsejet, over the course of one cycle. The blue line was a linear curve fit for the points on the vortex path when \(x/d\) was greater than one.

Scattered points in Figure 7-6 represent completed simulations for different geometries of the tapered ejectors. The base case is located on the linear curve fit for the vortex path. From the base case, three groups of results were studied. The first group contains ejectors at the same distance from the exit of the pulsejet, but with different diameters; the second group contains ejectors with the same diameter at different distances from the exit of the pulsejet; the third group contains ejectors with the distance and diameter staying on the linear curve fit line. Here, \(x/d\) is used to indicate the distance of the throat of the ejector from the exit of the pulsejet as shown in Figure 7-2.

Figure 7-6. Vortex ring path and simulated cases

Figure 7-7 shows thrust augmentation for ejectors with different \(D/d\) at the same distance downstream the exit of the pulsejet, namely, \(x/d=2.6\). Results indicate that thrust augmentation is significantly larger at the diameters near the base case. If the diameter of the
ejector is too small, it cannot capture the vortex. On the other hand, if the diameter of the ejector is too large, it allows too much of the vortex entering the ejector.

![Thrust augmentation as a function of D/d](image)

*Figure 7-7. Thrust augmentation as a function of D/d*

Figure 7-8 shows thrust augmentation versus distance from the exit of the pulsejet for different groups mentioned above. The red line represents ejectors of the third group mentioned above. The diameter of the ejector increases with the distance from the exit of the pulsejet. The results suggest that for \(x/d > 1.8\), thrust augmentation decreases with the increase of the distance from the exit of the pulsejet for the ejectors in the third group. For the ejectors in the second group, with a constant ejector diameter of \(D/d = 2.4\), there is a maximum value of augmentation attained when \(x/d = 2.2\). Further study also suggests that in each group, the largest augmentation occurs while the diameter of the ejector is a little bit larger than that of the vortex core. Results are shown in Figure 7-7 by a green line. The diameters of the three cases with \(x/d\) equal to 1.8, 2.2 and 2.6, respectively, are 1.25%, 1.95%
and 5.1% larger than that of the vortex core respectively. Compared with the cases in the third group with $x/d$ equal to 2.2 and 2.6, the thrust augmentations are 6% larger.

![Thrust Augmentation vs $x/d$](image)

**Figure 7-8. Thrust augmentation as a function of $x/d$**

### 7.3.2 Ejector length Effects

It is also observed that the pressure oscillations in the ejector are quite large, resulting in oscillations of thrust on the ejector. Figure 7-9 shows the pressure oscillations in the middle of the pulsejet, which displays interesting frequency characteristics. The major frequency was acquired from the pulsejet operation; a minor frequency was explained by the acoustic reflection at both ends of the ejector.
As discussed in Ref. [10], velocity in the ejector could be divided into steady component and unsteady component, larger unsteady component could lead to larger thrust. The frequency relationship between the ejector and the source will have a great effect on the unsteady oscillation in the ejector and thus affect the thrust augmentation. A frequency match between ejector and pulsejet was particularly interesting. Increasing or decreasing the ejector length could make its frequency match the frequency of the pulsejet. However, to make the frequency of the ejector equal to that of the pulsejet, a quite long ejector is needed, which makes it impractical. In this paper, ejector length was changed to make the acoustic frequency of the ejector 2 to 3 times as large as the source frequency. Total of five ejectors with different lengths were simulated, with frequency 2, 2.25, 2.5, 2.75 and 3 times as large as source frequency. The frequency for the base case is 2.5 times of the source frequency. Results were shown in Figure 7-10 and compared with reported values in Ref. [11]. The diameter of the simulation was not exactly the same as the reported value; $D/d$ was 2.4 for
simulation results but 2.46 for the results reported in Re. [11]. However, the simulation results suggest the same trend as in Ref. [11].

![Thrust Augmentation vs L/d](image)

*Figure 7-10. Thrust augmentation as a function of L/d*

This result indicates that the ejector length has a minor effect on thrust augmentation. Although there is significant acoustic oscillation in the ejector, but no sufficient evidence could support the thrust augmentation related to the frequency match.

### 7.4 Conclusions

From the analysis presented above, the following conclusions were obtained:

1. Consistent with experimental results, computational results predicted that tapered ejector generated a higher thrust augmentation than a straight ejector.
2. Performance of the ejector was related to the vortex generated at the exit of the pulsejet.
3. Ejector generated a higher thrust augmentation when the diameter of the ejector was closer to the diameter of the vortex core. Ejector with a little bit larger diameter than the vortex diameter provided the highest thrust augmentation.

4. Although high pressure oscillations were generated in an ejector for pulsed driving source, no sufficient evidence could support that the effect of the length of the ejector depended on the ratio of the frequency of the source and the ejector.
REFERENCES


APPENDIX
APPENDIX A: S-function in MATLAB for the 1-D pipe flow

function [sys,x0,str,ts]=limintm(t,x,u,flag,L,d0,ar,n,p0,t0)

R=8314/29;

gama=1.35;

dx=L/(n-1);

d1=[0:n-1];

d=d1*d0*(ar^0.5-1)/(n-1)+d0;

dt=0.5*dx/sqrt(gama*R*2200);

areax=3.142.*d.^2/4;

dave=(d(1:n-1)+d(2:n))/2;

areas=3.142.*dave*dx;

switch flag

% Initialization 

% case 0

[sys,x0,str,ts] = mdlInitializeSizes(p0,n,R,t0,dt);
%% % Update %

case 2

    sys = mdlUpdate(t,x,u,p0,t0,n,R,dx,dt,d,areax,areas,gama);

%% % Output %

case 3

    sys = mdlOutputs(t,x,u,n,gama,areax,p0,t0,R);

%% % Terminate %

case 9
sys = []; % do nothing

% Unexpected flags %

otherwise

    error([unhandled flag = ',num2str(flag)];)

eend

% mdlInitializeSizes
% Return the sizes, initial conditions, and sample times for the S-function.

function [sys,x0,str,ts] = mdlInitializeSizes(p0,n,R,t0,dt)
sizes = simsizes;

128
sizes.NumContStates  = 0;
sizes.NumDiscStates  = 3*(n-1);
sizes.NumOutputs     = 4;
sizes.NumInputs      = 2;
sizes.DirFeedthrough = 1;
sizes.NumSampleTimes = 1;
sys = simsizes(sizes);
str = [];
x0  = [p0.*ones(1,n-1) zeros(1,n-1) p0/(R*t0).*ones(1,n-1)]';
ts  = [dt 0];   % sample time: [period, offset]

% end mdlInitializeSizes
%
%=================================================================
% mdlUpdate
% Handle discrete state updates, sample time hits,
% and major time step requirements.

%=================================================================

129
function sys = mdlUpdate(t,x,u,p0,t0,n,R,dx,d,d,areas,gama)

    pc=u(2);       %input is combustion chamber pressure and temperature
    tc=u(1);
    twall=1000*exp((1:n-1)*log(0.4)/n);
    rhoc=pc/(R*tc); rho0=p0/(R*t0);
    x1=x.);
    p=[pc x1(1:n-1) 1];
    v=[0 x1(n:2*(n-1)) 0];
    rho=[rhoc x1(2*n-1:3*(n-1)) rho0];

    if v(2)>=0
        v(1)=min((2*v(2)-v(3)),v(2));
        rho(1)=0.1*(rhoc-rho(2))+rho(2);
        p(1)=pc-rho(1)*v(1)^2/2;
    else
        v(1)=min((2*v(2)-v(3)),v(2));
        p(1)=pc;
    end
rho(1)=rho(2);
end

if v(n)>=0
    v(n+1)=v(n);
    rho(n+1)=rho(n);
    p(n+1)=p0;
else
    v(n+1)=0.94*v(n);
    p(n+1)=p0-rho(n+1)*v(n+1)^2/2;
    rho(n+1)=0.1*(rho0-rho(n))+rho(n);
end

rl=rho(1:n);
vl=v(1:n);
pl=p(1:n);
hol=gama.*pl./((gama-1).*rl)+0.5.*vl.^2;
al=sqrt(abs(gama.*pl./rl));

rr=rho(2:n+1);

vr=v(2:n+1);

pr=p(2:n+1);

hor=gama.*pr./((gama-1).*rr)+0.5.*vr.^2;

ar=sqrt(abs(gama.*pr./rr));

asave=0.5.*(al+ar);

xml=vl./asave;

xmr=vr./asave;

all=0.5.*(1+sign(xml));

alr=0.5.*(1-sign(xmr));

btl=-max(0, 1-floor(abs(xml)));

btr=-max(0, 1-floor(abs(xmr)));
cvlplus = all.*(1+btl).*xml-btl/4.*(xml+1).^2;
cvlmins = alr.*(1+btr).*xmr+btr/4.*(xmr-1).^2;

fmplus = areax.*rl.*asave.*cvlplus;
fmmins = areax.*rr.*asave.*cvlmins;

dl = all.*(1+btl)-btl.*(xml+1).^2.*(2-xml)/4;
dr = alr.*(1+btr)-btr.*(xmr-1).^2.*(2-xmr)/4;
pave = dl.*pl+dr.*pr;

fv1 = fmplus+fmmins;
fv2 = fmplus.*abs(vl)+fmmins.*abs(vr)+areax.*pave;
fv3 = fmplus.*hol+fmmins.*hor;

%add friction
pin = p(2:n); rin = rho(2:n); vin = v(2:n);

din = (d(1:n-1)+d(2:n))/2;
temp=gama.*pin./rin/R;

niu=1.46e-6.*(twall.^1.5./(twall+111));

Red=abs(rin.*abs(vin).*din./niu)+1;

if (Red<2000)
    f=64./Red;
else
    f=0.316*Red.^-0.25/8;
end

tao=f.*rin.*vin.*abs(vin);

%add heat transfer


%prr=0.7;

nud=0.3+0.62*Red.^0.5*0.7^(1/3)/(1+(0.4/0.7)^(2/3))^(1/4).*sqrt(1+(Red/282000).^(5/8)).^0.8;

%Average Nu number

k=1.5207e-11*temp.^3-4.8574e-8*temp.^2+1.0184e-4*temp-3.9333e-4;

%thermal conductivity
q = nud.*k./din.*areas.*(temp-twall);

res1 = fv1(2:n)-fv1(1:n-1);
res2 = fv2(2:n)-fv2(1:n-1)+tao.*areas;
res3 = fv3(2:n)-fv3(1:n-1)+tao.*areas.*vin+q;

% correction

dv = dx.*(d(2:n).^2+d(1:n-1).^2+d(2:n).*d(1:n-1)).*3.14/12;

rin = rin-dt./dv.*res1;

rin = max(rhoc,rin);  % prevent density from being zero or negative

vin = vin-dt./dv.*res2./rin;

pin = pin-dt./dv*(gama-1).*(-vin.^2/2.*res1-vin.*res2+res3);

sys = [pin vin rin].';

% end mdlUpdate%
% mdlOutputs

% Return the output vector for the S-function

%=================================================================
% %
% function sys = mdlOutputs(t,x,u,n,gama,areax,p0,t0,R)
% %x(n) is v(1), x(2*n-1) is rho(1)
% pc=u(2);           %input is combustion chamber pressure and temperature
% tc=u(1);
% rhoc=pc/(R*tc);    %input is combustion chamber pressure and temperature
% rho0=p0/(R*t0);

% if (x(n)<0)
%     air = -x(2*n-1)*x(n)*areax(1);
% else
%     air=0;
% end

% t1 = x(n-1)/(R*x(2*n-1));  % use pressure divided by R and density to calculate temperature
sys = [areax(1)*x(n); air; t1;((x(n-1)-p0)+x(3*n-3)*x(2*n-2)^2)*areax(n)];

%x(n) is velocity to chamber, the first item provide the rate of volume enter and leave the combustion chamber, could be used to calculate the pressure difference and inlet state;

%x(n-1) is pressure at the opening, the third item provide p+rho*v^2, thrust.

% end mdlOutputs