ABSTRACT

SAYRES JR., JOHN SCOTT. Computational Fluid Dynamics for Pulsejets and Pulsejet Related Technologies. (Under the direction of Dr. William L. Roberts).

The pulsejet is one of the simplest forms of air-breathing propulsion ever developed. They are scalable, light weight, low cost, and fairly efficient at converting fuel to heat and thrust. This research effort focuses on the using mainly computational software to simulate and predict the working nature of the pulsejet.

There is an emphasis on expanding on previous research on 50 cm valved pulsejets and adapting this technology to smaller sizes of 25 cm and 12 cm. The performance of these pulsejets can be improved by the use of an augmenter and in this report there is a large study on how the free stream flight speed affects this performance improvement. Lastly, there is also an effort in modeling valveless pulsejets on the 5 cm to 25 cm scale.

The 50 cm augmented valved pulsejet obtains an augmentation ratio of 2.16 when running statically. This augmentation ratio drops when forward flight speed increases due to the increased drag that comes with a higher cross sectional area. The augmenter becomes essentially a net loss at speeds over 32 m/s. The 12 cm valved pulsejet is proven to be theoretically possible though it isn’t as fuel efficient as the 50 cm valved pulsejet with the average Isp dropping from ~2200 s at the 50 cm scale to ~1800 s at the 12 cm scale. Finally, there is some future work that appears to be an area that could be looked into to increase thrust further, such as the U-shaped pulsejet.
Computational Fluid Dynamics for Pulsejets and Pulsejet Related Technologies.

by
John Scott Sayres, Jr.

A thesis submitted to the Graduate Faculty of
North Carolina State University
in partial fulfillment of the
requirements for the degree of
Master of Science

Aerospace Engineering

Raleigh, North Carolina

2011

APPROVED BY:

_____________________________  ______________________________
Dr. William L. Roberts        Dr. Tarek Echekki
Committee Chair

_____________________________
Dr. Andre Mazzoleni
DEDICATION

This thesis is dedicated to my family; to my parents, John S. Sayres and Joyce A. Sayres, for whom without their continued support I would not have accomplished all that I have, and to my older sister, Jennifer A. Sayres, for whom always keeps my laughing and smiling through the years.
BIOGRAPHY

John S. Sayres, Jr. was born in Baltimore, Maryland on November 10, 1986. After spending several years in Maryland he and his family relocated to Greensboro, North Carolina in the fall of 1991. In Greensboro he spent his childhood with an academic fascination in science and math as well as several extracurricular hobbies including basketball, football, track, videogames, and music. He picked up the electric guitar in 2001 and has played in several local bands with an affinity for hard rock and heavy metal. He graduated from Northwest Guilford High School in May of 2004.

After graduating High School, John immediately enrolled at North Carolina State University majoring in Aerospace Engineering with a minor in Physics. John graduated cum laude with a Bachelor of Science in May of 2009 and followed immediately into the graduate school at NC State under the advising of Dr. William L. Roberts. After completing his Master of Science he plans to enter the Aerospace Engineering industry.
I would like to thank Dr. William L. Roberts for taking me on as a researcher and giving me guidance and knowledge. A large portion of the research was funded by Lockheed-Martin who helped me to get started on this research project. A large thank you goes to Dr. Fei Zheng for teaching me and mentoring me on the computational software used in this project. I would also like to thank my lab mates Joe Scroggins, Scott Steinmetz, Myles Bohon, Kim Bagian, and Nolan Cousineau for their help out at the AERL. Also, in addition to my parents and my sister, I would like to thank all of my band mates in Praetorius; Dan Nusz, Elliot Madre, Alex Kiser, Sam Defilipp, James Ross, Peter Lemieux, and Larson Kilstrom as well as Michael Hobbs for always keeping my sane during these hectic years of studying and researching in the field of engineering.
# TABLE OF CONTENTS

LIST OF FIGURES ..................................................................................................................... vii

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

CHAPTER 1 - Introduction ........................................................................................................... 1
  1.1 HISTORY .............................................................................................................................. 1
  1.2 THEORY AND JET DESIGN ................................................................................................. 4
  1.3 PREVIOUS RESEARCH ....................................................................................................... 8
    1.3.1 Experimental Research at NCSU .................................................................................. 8
    1.3.2 Computational Research at NCSU ............................................................................. 10
  1.4 GOALS OF THIS PROJECT ................................................................................................. 11

CHAPTER 2 – Computational Setup ............................................................................................... 12
  2.1 HARDWARE ....................................................................................................................... 12
  2.2 SOFTWARE ......................................................................................................................... 13
    2.2.1 Working with ANSYS ................................................................................................. 13
    2.2.2 Working with CFX ...................................................................................................... 16

CHAPTER 3 – Valveless Pulsejets................................................................................................. 18
  3.1 FUEL INJECTION ............................................................................................................... 18
  3.2 VALVELESS 12 CM PULSEJET ......................................................................................... 19
  3.3 INLET OPTIMIZATION ...................................................................................................... 23
  3.4 U-SHAPED PULSEJETS ..................................................................................................... 25
    3.4.1 U-Shaped Pulsejet Drag ............................................................................................. 26
    3.4.2 U-Shaped Pulsejet Thrust .......................................................................................... 28

CHAPTER 4 – Valved Pulsejets.................................................................................................... 32
LIST OF FIGURES

Figure 1-1: Simple valveless pulsejet in half section view ......................................................... 2
Figure 1-2: U-Shaped pulsejet ........................................................................................................ 3
Figure 1-3: V-1 “Buzz-Bomb” with Aurgus As 014 pulsejet powerplant ...................................... 4
Figure 1-4: General pulsejet design with flare .............................................................................. 5
Figure 1-5: p-V diagram for the Humphrey Cycle ........................................................................ 6
Figure 1-6: Running 25 cm pulsejet .............................................................................................. 9
Figure 2-1: 2-D ANSYS geometry .............................................................................................. 14
Figure 2-2: 3-D ANSYS geometry .............................................................................................. 14
Figure 2-3: CFD Wind Tunnel dimensions .................................................................................. 15
Figure 2-4: Valved pulsejet with augmenter mesh ...................................................................... 16
Figure 3-1: Symmetric four hole fuel injector ............................................................................ 19
Figure 3-2: Valveless 12cm pulsejet with varying combustion chamber to exhaust tube lengths ........................................................................................................................................ 20
Figure 3-3: Varying transition length data plotted with trend line .......................................... 21
Figure 3-4: 25 cm variable inlet valveless pulsejet ...................................................................... 23
Figure 3-5: 25 cm variable inlet valveless pulsejet thrust with trend line ................................ 24
Figure 3-6: 31cm U-shaped jets. No aerodynamic cone (top) and with aerodynamic cone (bottom) .................................................................................................................................................... 27
Figure 3-7: 28cm U-shaped jet enclosed in smooth NACA 0012 Airfoil .................................. 28
Figure 3-8: 25cm to 31 cm U-shaped jets ..................................................................................... 29
Figure 3-9: 25cm U-shaped chamber pressure (top) compared with 31 cm U-shaped chamber pressure (bottom) ................................................................................................................................................................................................. 30

Figure 4-1: 12 cm 2-D valved pulsejet. Valve location highlighted in red................................................. 34

Figure 4-2: Sickle valve with fuel injection points (shown in red)............................................................. 37

Figure 5-1: Running 50cm, valved, pulsejet with augmenter ................................................................. 40

Figure 5-2: Augmenter location as prescribed by Fei Zheng ................................................................. 40

Figure 5-3: 50 cm valved pulsejet with augmenter half-section view .................................................. 42

Figure 5-4: 50 cm valved pulsejet with augmenter CAD model .......................................................... 43

Figure 5-5: Total thrust vs free stream velocity ....................................................................................... 46

Figure 6-1: Wind tunnel with pulsejet on sting and dynamometer....................................................... 50

Figure 6-2: Close up on dynamometer ................................................................................................. 51

Figure 6-3: CFD 6” wind tunnel with 12 cm pulsejet on NACA 0012 sting........................................ 52

Figure 6-4: Drag versus free stream velocity, computational drag compared with experimental drag................................................................................................................................. 54

Figure 6-5: Experimental cold flow testing rig courtesy of Joe Scroggins........................................ 56

Figure 6-6: CFD cold flow testing device ............................................................................................. 57

Figure 6-7: Cold flow testing at 1000 Hz for various high pressures .................................................. 58
LIST OF TABLES

Table 3-1: Varying transition length data ................................................................. 22
Table 3-2: Coefficient of drag on the u-shaped jet ................................................... 27
Table 3-3: U-Shaped pulsejet thrust at various lengths ........................................... 31
Table 4-1: Equivalence ratio performance results .................................................... 35
Table 4-2: Sickle valve performance results at three fuel injection rates ................. 38
Table 5-1: Drag on augmenter alone with increasing free stream velocity ............. 44
Table 5-2: Performance of 50 cm valved pulsejet with augmenter .......................... 45
Table 5-3: Thrust measured at the exit of the pulsejet with augmenter .................. 47
Table 6-1: Drag and Cd for 12 cm jet on NACA 0012 sting in 6” CFD wind tunnel .... 53
Table 6-2: Average mass flux and necessary high pressure chamber volume at 1000 Hz ..... 59
CHAPTER 1

Introduction

1.1 History

The pulsejet is one of the simplest forms of propulsion known to man. They are known for having little to no moving parts, scalability, relatively low cost, ease of use, and extremely high noise. Pulsejets at the simplest level work by means of pulsed combustion and were invented in the early 1900s.

Pulsejets generally operate either valvelessly, in which the combustion chamber is always open to the free stream, or valved in which the inlet is separated from the combustion chamber by some sort of mechanical valve. The modern valveless pulsejet was created in the early twentieth century by Marconet as well as Schmidt. This simple propulsion device consisted of an open inlet leading into a combustion chamber and exhausting out of the back by an often flared exhaust tube. The main drawback of this simple design is that, during exhaling cycles, there is a significant loss of products out of the front of the jet. Studies have shown that as much as a third of the products of combustion leave through the front, causing a large amount of negative thrust [1]. A CAD mockup of this simple design can be seen below in Figure 1-1.
An improvement on the valveless jet was made by Ray Lockwood of Hiller Aircraft Corporation in the 1960’s by creating a “U-shaped” pulsejet in which the inlet and the exhaust tube are both facing the same direction. This allows all thrust to be facing the same direction, maximizing thrust as shown in Figure 1-2 [2]. This U-shape has the obvious advantage of maximizing the thrust but also has several drawbacks. Possibly the largest is that they have nearly double the cross sectional area of a traditional straight pulsejet. This coupled with the fact that drag increases with the square of the velocity, creates a hefty drag penalty at higher forward flight speeds. Nonetheless this U-shaped design is still very prominent among hobbyists that still tinker with pulsejets today.
Perhaps the most widely known use of pulsejets came to prominence during World War II with Germany’s V-1 “Buzz-Bomb” shown in Figure 1-3. The V-1 was powered by the Argus As 014 pulsejet. This jet was made from steel and got its nickname due to its operational frequency of about 45 Hz. This produced a loud, low pitched buzz when flying. The As 014 was capable of producing approximately 2700 Newtons of thrust. However, due to the pulsejets inherent low thermodynamic efficiency the V-1 was limited to a range of approximately 150 to 250 miles. The As 014 was the first pulsejet ever to be put into mass production, however there were only approximately 30,000 of these pulsejets made [3].
Nowadays the pulsejet is widely known as more of a hobby for many “garage engineers.” Due to the high heat output in noises on the order of 100 dB pulsejets aren’t very practical for commercial propulsion use. But, they still exist in the realms of experimental analysis, and it is off of these many designs that the research at NC State has been based for the past decade. This project will delve into what does and doesn’t work with these designs and how to best improve upon them.

1.2 Theory and Jet Design

The operation of the pulsejet has been fairly well known for nearly a century. Because the jet has few moving parts it is almost completely described by its acoustic properties throughout the jet. There has been a large amount of computational and experimental research done at NC State over the past decade and this paper builds upon that research.
The main design of any pulsejet consists of three main sections as shown below in Figure 1-4. The inlet is a round tube that serves as the main method of inhaling fresh air into the system on each cycle. It is generally the shortest section of the pulsejet and, thus, also has the smallest cross sectional area. After the inlet there can either be valves, which cyclically restrict mass from flowing back through the inlet, or the jet can operate “valvelessly” in which there is no restrictor. Next, the inlet leads into the combustion chamber. The combustion chamber generally has the largest cross sectional area and serves as a sort of stagnation zone for the fresh air to mix with injected fuel for combustion. The inlet with the combustion chamber can be thought of as a Helmholtz resonator. The combustion chamber then leads into the exhaust tube, which can be thought of as a simple wave tube. The exhaust tube is generally much longer than the inlet and thus has a slightly bigger cross sectional area, not to exceed that of the combustion chamber. This exhaust tube can also have a flare on the end to help with vortex generation and direction.

Figure 1-4: General pulsejet design with flare
Pulsejets operate on the Humphrey cycle as shown in Figure 1-5. The four step cycle starts at state 1 with isentropic compression followed by isochoric heat addition, isentropic expansion, and lastly, isobaric heat rejection as it returns to state 1.

Figure 1-5: p-V diagram for the Humphrey Cycle

The general design for a valveless pulsejet can be approximated by matching the frequency of a short length Helmholtz resonator for the inlet and combustion chamber to a long length Helmholtz resonator for the outlet and combustion chamber [4]. This allows for the design of the pulsejet to come from the following two equations [5]:

\[
\begin{align*}
\frac{\pi}{2} & = \frac{1}{\sqrt{\alpha_1 \cdot \alpha_2}} \\
\frac{\pi}{2} & = \frac{1}{\sqrt{\alpha_2 \cdot \alpha_3}}
\end{align*}
\]
These symbols are representative of the following values where $i$ describes the inlet properties and $e$ describes the exhaust tube properties:

- $f$  Frequency
- $c$  Local Speed of Sound
- $S$  Cross-Sectional Area
- $L$  Length
- $V$  Combustion Chamber Volume

By setting the frequency of the inlet to the frequency of the exhaust tube the jet is “fully resonant” meaning that the inlet and the exhaust tube are resonant with each other and the jet will operate under self sustaining combustion. The equation for the inlet frequency can be solved explicitly. The frequency for the exhaust tube, however, must be solved implicitly due to the presence of the frequency in the tangent coefficient as well as inside the tangent operator. When designing the jets for this project a simple Microsoft Excel solver was created to solve for the frequencies and match them based on input parameters of overall
length, desired radius of the exhaust tube, and the ratio of the inlet volume to the combustion chamber volume. Once all of the parameters of the jet are pinned down a model can be created using CAD software.

1.3 Previous Research

1.3.1 Experimental Research at NCSU

At NCSU there has been a large amount of research conducted in experimentally running pulsejets. Previous researchers include Michael Schoen, Rob Orden, Christian McCaulley, Ranjith Kumar, Joe Scroggins, and Scott Steinmetz. Much of this research involved looking at overall lengths, areas, and geometries of the jet to try and understand how varying these parameters affect the operation of a running pulsejet. There have been numerous studies on pulsejets from 12 centimeters up to 1 meter in overall length. Figure 1-6 shows a 25 centimeter pulsejet with an exhaust flare and variable inlet length running on a Propyne fuel source.
Michael Schoen found that a jet’s operation is mainly characterized by the relationship between the length of the jet and the kinetics of the fuel as well as the relationship between the cross sectional areas of the inlet to the combustion chamber [7]. Rob Orden looked at 50 centimeter pulsejets and found a relationship between the Helmholtz frequency of the inlet and the wave tube frequency of the outlet. Matching these frequencies is crucial to jet operation and is this frequency matching is used in designing the jets that are simulated computationally in this study [8]. Christian McCaulley looked at the effects of various fuels on a 25 centimeter pulsejet. He, as well as several other researchers, found that the method of fuel injection is very important in starting a pulsejet. There is a “sweet spot”
located about a third of the way into the combustion chamber where the fuel should be injected. There have also been studies on the direction of fuel injection which include injecting axially down the centerline or in the radial direction from the centerline out to the combustion chamber walls [9].

Joe Scroggins also participated in understanding how “augmenters” can help increase net thrust on a static jet. Experimentally he found that an augmenter can produce a net increase in thrust by up to a factor of 2 when run statically. Scott Steimnetz performed a fuel study and found that these fuels have varying performance based on their kinetics. A study run concurrently with my project is looking at how heating the fuel can allow a jet to run when it previously would not run on the unheated fuel because the kinetics were not fast enough. This has lead to the selection of Propane as the fuel of choice for jets above 12 centimeters and Hydrogen for smaller jets.

1.3.2 Computational Research at NCSU

There has also been a considerable amount of effort in coming up with numerical models for simulation of these pulsejets at NCSU. Tao Geng, Fei Zheng, and Todd Travis have all conducted simulations to describe the flow properties of the pulsejet. Tao Geng studied the overall fluid mechanics of the running pulsejet, the effects of the starting vortex at the pulsejet exhaust tube, and flare designs. Fei Zheng studied the effects of objects in a valveless combustion chamber, the effects of the location of the augmenter on valved jets, as well as creating a MATLAB code to optimize jet geometry and the FORTRAN subroutine used to model valve dynamics for CFD simulations of valved pulsejets. Todd Travis has
studied constant volume combustion as well as valved pulsejets. I have conducted experiments on many of these previous areas of research in hopes of expanding our knowledge of these pulsejets.

1.4 Goals of this Project

As previously mentioned, this project is mainly studying the workings of valved and valveless pulsejets using numerical computer simulations. There has been a large amount of previous experimental and computational research done so this project mainly aims at using the computer to predict behavior of areas previously not studied. This includes looking at the effects of forward flight speed on valved pulsejet augmentation, the effects of 3-dimensional fuel injection, wind tunnel drag, and geometry optimization. There is also a look at trying to simulate the u-shaped valveless pulsejet to effectively eliminate all negative thrust.
CHAPTER 2

Computational Setup

2.1 Hardware

Because this research project has a heavy basis on computational fluid dynamics the majority of the research was done on computers. Three main computational devices were used for this project. While I was learning how to build CFD simulations, early testing was done on the NCSU IBM Blade Center which is powered by a 3.0 GHz Intel Xeon processor. This blade center is known as High Performance Computing, or HPC.

The majority simulations presented in this report were split between two other Dell workstations. One workstation housed a dual quad-core 2.4 GHz Intel Xeon processor and 24 GB of RAM while the second workstation housed a dual quad-core 2.33 GHz Intel Xeon processor 16 GB of RAM. Because these machines each contained eight physical CPUs I was able to run up to eight separate CFD jobs at once. This is beneficial because the run time could vary anywhere from a few hours up to several weeks. For the average pulsejet simulation a single cycle takes between 12 to 24 hours depending on complexity.
2.2 Software

The hardware used for all computing was backed up by a variety of strong commercial engineering packages. The CAD modeling was done using either SolidWorks 2009 or ANSYS 11 Workbench. All of the CFD was done on CFX 11 using its -pre, -solver, and -post functionality. For valved pulsejet simulations the valve dynamics were calculated using a FORTRAN subroutine.

2.2.1 Working with ANSYS

After a CAD model for the pulsejet, or whatever object a numerical simulation will be performed on, a geometry and mesh file must be created. This is done in ANSYS workbench. For simulating a pulsejet I created a wind tunnel based on the size of the jet. A straight pulsejet can be modeled as an axisymmetric flow regime so a 2-D simulation is usually the easiest to model. This is achieved by creating a 4° wedge and then multiplying any area based numbers by a factor of 90. This decreases the number of calculations done by the computer for each iteration. For more complex geometries a 3-D tunnel is necessary and thus no simplifications are made. Figures 2-1 and 2-2 show the 2-D and 3-D geometry file for a pulsejet.
Figure 2-1: 2-D ANSYS geometry

Figure 2-2: 3-D ANSYS geometry
Due to the nature of the simulations being run in a wind tunnel it is necessary to take into account any boundary and wall effects that wouldn’t be seen in free stream conditions. The wind tunnel boundaries are generally created with sufficient distance from the jet that any boundary effects are negligible [6]. This is shown in Figure 2-3 where the boundaries are a fixed distance from the jet based on the jet inlet and outlet sizes.

![Figure 2-3: CFD Wind Tunnel dimensions](image)

In this case the dimensions are given as a factor of the di, the inlet diameter, or do, the exhaust tube diameter. These are chosen to reduce any wall effects such as reflected compression or expansion waves.

After the geometry file is made it is necessary to create a mesh for the solver to calculate all of the flow conditions. This is also done in ANSYS. For 2-D cases a single layer is used for the thickness of the wedge. Inflation is used for all cases to help accurately measure the conditions near the walls of the pulsejet. As seen in Figure 2-4 the mesh is much finer in points of interest, such as inside the jet and along the walls of the augmenter.
A typical 2-D mesh contains around 12,000 mesh cells whereas a 3-D mesh could contain 500,000 to 1,000,000 mesh cells. Once the mesh is created I can then begin to create the definition file to run the simulation in CFX with.

### 2.2.2 Working with CFX

The mesh file created in ANSYS is used to create a CFX definition file. The combustion is modeled using a single step eddy dissipation technique. Turbulence is modeled using the k-epsilon model. This model has been shown to have good accuracy as well as being scalable to a wide variety of problems [6].

The fuel flow rate and tunnel inlet speed are also defined here. In the presented simulations all of the flows are somewhere between 0 and 60 m/s and have constant free stream conditions. The fuel involved in the pulsejet simulations is stoichiometric propane at room temperature. In addition to these it has been seen that there is a large amount of heat lost through the walls of the jet. This is offset in the simulation by adding in a piecewise
function that keeps the combustion chamber at a constant temperature of 1000 K while the walls of the jet are linearly reduced to 400 K [6].

CFX also has a very simple way for monitoring flow properties as they are calculated by the solver. The user can place monitors in Cartesian space and dictate what he wants outputted. This can be local temperature, average pressure, area, etc. In a mesh consisting of thousands to millions of cells it is often more convenient to collect data through these monitor points as opposed to trying to track the flow properties at every mesh cell.

Lastly, for simulating valved pulsejets, a valve dynamics code was written in FORTRAN by Dr. Fei Zheng. This program is used for all of the valved testing done during this project. CFX has an area to input the directory of the compiled FORTRAN code and it can execute it in real time with the flow solver. This code can be seen in the Appendix.
CHAPTER 3

Valveless Pulsejets

As seen previously in Figure 1-4 the valveless pulsejet has an extremely simple design. The lack of any moving parts makes the design easy to build and also keeps costs down. CFD analysis was done on a variety of valveless pulsejets in the 12 cm to 25 cm range. There has been a considerable amount of research in this area but there are still a few areas that weren’t understood as well as others.

3.1 Fuel Injection

Generally a valveless pulsejet is started by injecting fuel in the combustion chamber at about a third of the way into the combustion chamber, as seen in previous research [6], and igniting with a spark plug. In CFX it is very complex to model an actual fuel injector inside of the running pulsejet. This necessitated a way to simplify the computer simulation while keeping the model accurate. The easiest way to do this is to introduce a point fuel source at the desired location and define a mass flux and velocity. We know the experimental mass flux based on our flow meters in the lab. The velocity can be calculated by hand based on conservation of mass. I modeled a fuel injector in CFX to see if this was an accurate way to inject the fuel. Figure 3-1 shows this modeled fuel injector. It is a four hole fuel injector with symmetry yielding two holes spraying fuel in opposite directions.
This simulation was done with imposed symmetry along the axis of the fuel injector. This leads to the design shown with two holes marked by yellow crosses as seen above. This is an exact replica of the experimental fuel injector which runs Hydrogen at 15.5 L/min. Simulating this flow of Hydrogen at the experimental flow rate yields flow velocities of 135 m/s on the top hole and 115 m/s on the bottom hole. These velocities along with the known mass flow rates are what are used for the simulations for the valveless pulsejets.

### 3.2 Valveless 12 cm Pulsejet

Based on scaling analysis it is thought that a 12 cm pulsejet should produce anywhere between 60 and 90 mN of thrust [8]. When looking at the equations above for designing the
pulsejet there is nothing that dictates the length of the transition zone between the combustion chamber and the exhaust tube. Tests were done with varying transition zone lengths to find the optimal design. Figure 3-2 shows the differing designs with the varying transition lengths highlighted in red. It is clear to see that the Inlet and combustion chamber lengths are kept consistent through these tests. The pulsejet is designed by convention to have a smooth transition into the exhaust tube; it is this transition that is being studied.

*Figure 3-2: Valveless 12cm pulsejet with varying combustion chamber to exhaust tube lengths*

The baseline for the transition length was arbitrarily set at 0.295 cm. Tests were run with transition lengths varying from 0.183 cm to 1.467 to determine if there is an
effect on performance. Figure 3-3 shows the conclusive results of these tests. There is definitely a trend on how the transition length affects performance of the jet. The optimal transition length appears to exist in the 0.45 cm to 0.55 cm range. By increasing the transition length from 0.295 cm to 0.492 cm the thrust increases 13% from 62.7 mN to 70.8 mN. A list of the data as the transition length increases can be seen in Table 3-1.

![Figure 3-3: Varying transition length data plotted with trend line](image-url)
### Table 3-1: Varying transition length data

<table>
<thead>
<tr>
<th>Transition Length (cm)</th>
<th>Thrust (mN)</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.183</td>
<td>55.6</td>
<td>1158</td>
</tr>
<tr>
<td>0.295</td>
<td>62.7</td>
<td>1144</td>
</tr>
<tr>
<td>0.385</td>
<td>68.2</td>
<td>1149</td>
</tr>
<tr>
<td>0.492</td>
<td>70.8</td>
<td>1149</td>
</tr>
<tr>
<td>0.621</td>
<td>67.5</td>
<td>1137</td>
</tr>
<tr>
<td>0.752</td>
<td>52.7</td>
<td>1141</td>
</tr>
</tbody>
</table>
3.3 Inlet Optimization

The 25 cm valveless pulsejet designed by Christian McCaulley had a unique design that allowed the inlet to be slightly modular in length. This inspired me to do a few tests based on how the inlet is designed and how to best optimize it. Figure 3-4 shows this 25 cm design. We know that the best operating pulsejet has its inlet frequency matched to its exhaust frequency [5]. I ran two sets of tests to test this theory. One involved keeping the inlet frequency the same and varying its length, while the other test involved keeping its volume the same and varying its length. Overall the inlet length was varied from 3 cm to 4.5 cm leaving the jet as 25 cm to 26.5 cm.

![Figure 3-4: 25 cm variable inlet valveless pulsejet](image)

It is important to note that the second set of tests of constant volume will yield varying frequency. Because the jet’s operation is very closely linked to frequency matching it should be expected that the further away the inlet frequency is from the exhaust tube frequency the poorer the jet’s performance should be. The results of this test are shown below in Figure 3-5.
As expected the constant volume tests with varying frequency fell off considerably with the lower lengths. This frequency was at nearly 800 Hz where the expected frequency should be at around 700 Hz. Unexpected, however, were the variations in the constant frequency tests. There is a general trend with a peak at around the 3.8 cm length, corresponding to an inlet radius of 0.37 cm. Where the two graphs cross indicates the baseline model that was studied by McCaulley. It shows that there are a couple of ways that the jet could be optimized and the best seems to be keeping the frequency at 700 Hz and at a length of 3.8 cm.
3.4 U-Shaped Pulsejets

As described in the introduction, the valveless pulsejet has the major drawback of exhausting nearly a third of its products out of the front. The Hiller Aircraft corporation helped to solve this problem by introducing the U-shaped pulsejet which faces the exhaust and the inlet in the same direction. This is the standard shape used by hobbyists for larger valveless pulsejets today. There has not been much research in these U-shaped jets at NCSU so I have done some simulations to try and get a sense of how these jets work. The main things that were studied were the effect of the bend on the fluid dynamics, the thrust increase, and the jet’s overall ability to function. A simple U-shaped pulsejet was shown above in Figure 1-2 and was the basis for this experiment.

The main benefit of the U-shaped pulsejet is to exhaust all of the products of combustion in the same direction. This shape also has the added benefit of allowing the user to increase the overall length of the jet while keeping a smaller form factor. For the U-shaped jets I tested the jets had an overall jet length of about 25 cm to 34 cm but the tip to bend length was still only about 12 cm. This is very helpful if the user is trying to install one of these jets into a small location. Increased length on a pulsejet generally leads to higher thrust. The benefits of having increased length are, however, offset by the increased cross sectional area, leading to a higher drag penalty. These tests were investigating the effects of having a U-shaped jet.

Because the jet is no longer axisymmetric these tests had to be done in full 3-D with approximately 600,000 mesh cells. This led to an increase in the computational time by a
factor of about 1.8. This increase in computational time led to tests taking several weeks to run and thus these tests were more proof of feasibility.

3.4.1 U-Shaped Pulsejet Drag

I started by investigating the effects of increased drag on the pulsejet as shown in Table 3-2. I did these tests on three different curved jets. There was a jet that was just our standard pulsejet but curved. I then took that same jet and added an aerodynamic cone to the nose of the jet and tested its drag. Lastly, I took the curved jet and enclosed it in a smooth airfoil based on the NACA 0012 design. These three jets are shown in Figures 3-6 and 3-7. The radius of curvature was decided upon as 1.5 times the outer diameter of the exhaust tube. This was done with manufacturing in mind to reduce what is known as pipe wrinkle. The curved jet enclosed in the smooth airfoil has its center of curvature located slightly off center of the actual extruded airfoil itself. This was done with the often-stated expectation that the jet will exhaust one-third of the products of combustion out of the inlet and two-thirds out of the exhaust tube. With this in mind the jet was placed with the exhaust tube placed closer to the center of the airfoil in order to eliminate any moment on the construct. Any curved pulsejet design will have to keep this in mind in order to reduce the amount of internal stresses that could possibly be a cause of failure in the design.
Table 3-2: Coefficient of drag on the u-shaped jet.

<table>
<thead>
<tr>
<th>Aerodynamic Property</th>
<th>Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard curved jet, no nose cone</td>
<td>0.347</td>
</tr>
<tr>
<td>Standard curved jet, with nose cone</td>
<td>0.285</td>
</tr>
<tr>
<td>Standard curved jet in smooth airfoil</td>
<td>0.249</td>
</tr>
</tbody>
</table>

Not surprisingly, the Cd drops with the addition of increased aerodynamics. What is interesting to note, however, is that the U-shaped jet enclosed in the smooth airfoil actually has a lower coefficient of drag than the standard straight jet. This suggests that it may be beneficial to look into enclosing a straight jet in some sort of smooth airfoil in the future.

Figure 3-6: 31cm U-shaped jets. No aerodynamic cone (top) and with aerodynamic cone (bottom)
3.4.2 U-Shaped Pulsejet Thrust

The transient combustion runs on the U-shaped pulsejets created many unexpected results. All of the following tests were run in a 25 m/s convective stream. The fuel flow rate was fairly high at about 0.24 g/s of pure propane. The tests began on a 25 cm curved pulsejet. After about 12 cycles the chamber pressures where still not quasi-stable. They were more random and not very even. This was very unexpected because the standard straight pulsejets are expected to have very consistent and quasi-stable chamber pressures. Because the 25 cm pressures were so unstable I surmised that the bend in the jet somehow alters the fluid mechanics as well as the combustion, thus throwing off the frequency matching. I took the same jet and simply extended the exhaust tube length after the bend up to an overall jet length of 28 cm and 31 cm. These three jets are shown in Figure 3-8. This led to better stabilization of the chamber pressures. The 25 cm chamber pressure compared
with the 31 cm chamber pressure is shown below in Figure 3-9. The 31 cm chamber pressure profile is much smoother and reaches the quasi-stable peak pressures as expected.

*Figure 3-8: 25cm to 31 cm U-shaped jets*
The overall thrust generated by these three curved jets is shown below in Table 3-3. In previous research we have learned to expect the valveless 25 cm pulse jet to produce thrust at approximately 400-500 mN. Because the straight jets exhaust a third of the products out of the front and two thirds out the back this leaves only a net of one third of the jets capable thrust being produced. The curved jets producing thrust on the order of 1.3 N to 1.5 N is thus very promising because it shows that the lost thrust is recovered while possibly being increased due to its increased exhaust tube length. Also, with its compact overall length of
approximately 12 cm the thrust is substantially better than the expected 90 mN of thrust on a straight 12 cm jet.

<table>
<thead>
<tr>
<th>Jet Length (cm)</th>
<th>Thrust (N)</th>
<th>Frequency (Hz)</th>
<th>Isp (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>1.351</td>
<td>697</td>
<td>574.4</td>
</tr>
<tr>
<td>28</td>
<td>1.497</td>
<td>649</td>
<td>636.5</td>
</tr>
<tr>
<td>31</td>
<td>1.539</td>
<td>634</td>
<td>654.3</td>
</tr>
</tbody>
</table>

*Table 3-3: U-Shaped pulsejet thrust at various lengths*
CHAPTER 4

Valved Pulsejets

Valved pulsejets have the unique ability to eliminate the release of combustion products out of the front of the pulsejet. This is due to the nature of the valve mechanics themselves. A valved pulsejet places a mechanical valve directly in between the inlet and the combustion chamber. This valve opens into the combustion chamber via some sort of hinging mechanism. This means that when the combustion chamber pressure is larger than atmospheric, as in the combustion phase, the valve will be forced shut but when it is less than atmospheric, as in the inhalation phase, it will be open. Thus, a properly designed valve will only be open when the jet is inhaling and no products of combustion will leave the combustion chamber, greatly increasing thrust.

In general there are two types of valve dynamics mechanisms used in CFX simulations, the linear spring model and the nonlinear spring model. The linear spring model is based on simple mass on a spring dynamics as in Hooke’s Law. This is simple, easy to manipulate in the computer, has many design options, and is proven in simulation. In this model the user simply inputs the inlet length, valve area, valve mass, and spring constant. The FORTRAN code written by Fei Zheng takes this data as well as the current chamber pressure, ambient pressure, and current simulation timestep and calculates the mass flux through the inlet. It has been shown in previous research that for the linear spring model the
valved pulsejet performance is maximized when the valve natural frequency is slightly above the operating frequency of the pulsejet [6].

The other type of valve dynamics mechanism is the nonlinear spring which is similar to the linear valve except its valve dynamics aren’t based on the linear Hooke’s Law but rather any number of other non linear dynamics. This model is much more difficult to implement, has not been proven in simulation and is thus not presented in this project.

There are some drawbacks to the valved pulsejet when compared to valveless pulsejets. They are slightly more complex than the valveless design and thus harder to implement. They are also less scalable; experimentally there have been working valveless pulsejets down to the scale of a few centimeters where as the valved pulsejet has proven incredibly difficult to get resonating at sizes less than 50 cm. Nevertheless a functioning valved pulsejet provides much improved thrust over the valveless pulsejet and is thus worth investigating.

4.1 Fuel-Air Mixture Ratio Effect

For the majority of these valved tests the simulation takes the pulsejet geometry and injects premixed fuel and air at the specified mass flow dictated by the FORTRAN code. What’s important to note is that this valve geometry takes the standard 12 cm pulsejet geometry and essentially eliminates the traditional pulsejet inlet and replaces it with the FORTRAN mass flow at the inlet of the combustion chamber. This valve is sized as the radius of the inlet. This is done as a simplification because it is assumed that no fluid flows
backwards out of the combustion chamber and through the inlet. Figure 4-1 shows this valve geometry configuration.

![Figure 4-1: 12 cm 2-D valved pulsejet. Valve location highlighted in red.](image)

In CFX you can take this fuel flow rate and specify the equivalence ratio, $\varphi$, for the fuel/oxidizer ratio. This is important because it tells you how much fuel-air you are using in relation to the stoichiometric amount. The stoichiometric amount is determined by the atom balance on the chemical equation for combustion of propane, which leads to the mass of fuel to mass of oxidizer ratio to be equal to 0.275. For atmospheric conditions the mass fraction of Oxygen is about 0.232 and knowing this the mass fraction of fuel for the equivalence ratio can be solved for. The equations for equivalence ratio and propane combustion are shown below.

$$\varphi = \frac{m_{fuel}/m_{ox}}{(m_{fuel}/m_{ox})_{st}}$$

$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$
I designed a set of tests to be conducted to find the effect of running a valved pulsejet lean or rich. These tests were done on a 12 cm valved pulsejet in a 25 m/s convective stream. These tests were conducted with propane fuel at equivalence ratios of 0.7-1.1 where 1.0 is stoichiometric. The results from these tests are shown below in Table 4-1.

Table 4-1: Equivalence ratio performance results

<table>
<thead>
<tr>
<th>$\varphi$</th>
<th>0.7</th>
<th>0.8</th>
<th>0.9</th>
<th>1.0</th>
<th>1.1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel Flux (g/s)</td>
<td>0.0167</td>
<td>0.0192</td>
<td>0.0213</td>
<td>0.0240</td>
<td>0.0258</td>
</tr>
<tr>
<td>Thrust (mN)</td>
<td>335.1</td>
<td>370.9</td>
<td>391.3</td>
<td>423.9</td>
<td>432.1</td>
</tr>
<tr>
<td>Isp (s)</td>
<td>2048.3</td>
<td>1966.6</td>
<td>1870.6</td>
<td>1803.3</td>
<td>1708.4</td>
</tr>
<tr>
<td>Frequency (Hz)</td>
<td>862.1</td>
<td>876.4</td>
<td>879.9</td>
<td>885.3</td>
<td>886.1</td>
</tr>
</tbody>
</table>

By noting the specific impulse, Isp, it is easy to see that jet performance is definitely increased by running the jet lean, but this is at the expense of losing net thrust. Not surprisingly the general trends are that thrust and fuel consumption both increase with running the jet richer, while the specific impulse decreases with increased equivalence ratio. While these effects aren’t surprising they are still important. For example, if your goal is to create a pulsejet that aim to have a specific impulse of approximately 1800 seconds it could be achieved by running lean, thus saving fuel.
4.2 Sickle Valve 3-D Simulations

The 2-D simulations provide a good first step at simulating the valved pulsejet performance but they are quite different than the actual running pulsejet geometry. It isn’t very accurate to model the valve as simply the exact size of the valveless inlet with premixed fuel and air as the injected mass. To increase the fidelity of the simulations I’ve modeled a three dimensional valve created by Nolan Cousineau, known as the “Sickle Valve” which helps to increase mixing and swirling in the combustion chamber.

As seen in Figure 4-2 the sickle valve takes two curved valves that open horizontally on opposing hinges, which create swirling airflow into the combustion chamber. The Fuel, propane in this case, is then injected at a point 30° off center and then at a 30° angle towards the opposite wall at 130 m/s. This is done to maximize the mixing with the swirled air as it enters the combustion chamber. The test results are shown below in Table 4-2.
In the sickle valve model the fuel and air is not premixed. To achieve this I run the
FORTRAN subroutine with the specification that the mass flow from the valve consist of air
with a mass fraction of 0.232 Oxygen with the remaining mass as Nitrogen. The fuel is then
injected as seen above in Figure 4-2, thus modeling the experimental sickle valve more
closely.

From the 2-D valved simulations it was shown that at stoichiometric mass flow
conditions propane was being injected with a mass flow of 0.024 g/s. I thus conducted three
tests, a lean fuel run at 0.020 g/s, a stoichiometric fuel run at 0.024 g/s, and a rich fuel run at
0.028 g/s.
Table 4-2: Sickle valve performance results at three fuel injection rates

<table>
<thead>
<tr>
<th>Fuel Flow (g/s)</th>
<th>0.020</th>
<th>0.024</th>
<th>0.028</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thrust (mN)</td>
<td>240</td>
<td>260</td>
<td>280</td>
</tr>
<tr>
<td>Isp (s)</td>
<td>1280</td>
<td>1160</td>
<td>1080</td>
</tr>
<tr>
<td>Frequency (Hz)</td>
<td>1010</td>
<td>1011</td>
<td>1016</td>
</tr>
</tbody>
</table>

From these sickle valve tests it is first very apparent that the fuel-injected case performs much less favorably than the premixed case. The specific impulse is thus much lower. What is interesting is that the frequency is actually much higher and more constant as opposed to the premixed case, which seems to increase more drastically with higher fuel consumption.
CHAPTER 5

Augmenter Effects

One way to increase thrust from the pulsejet is by implementing a device known as an augmenter. This device is placed behind the exhaust tube at the back of the pulsejet as seen below in Figure 5-1. The figure depicts a running, 50cm, valved pulsejet with augmenter. This setup was completely designed by Joe Scroggins. This device works by utilizing the vortex off of the end of the flare of the exhaust tube to create a low pressure zone on the inside of the lip of the augmenter. This incites entrainment of mass, pulling ambient air into the augmenter and ejecting it out of the exit. This entrainment of mass increases thrust by playing on Newton’s Second Law by increasing mass flux. The drawback of increasing the mass flux is that there is a trade off of decreased exit velocity. A well-designed augmenter, however, successfully balances this trade off by increasing mass flux more than the decrease in exit velocity [6].

Fei Zheng studied this augmenter design extensively and his optimized augmenter geometry is shown below in Figure 5-2. This design is defined by the augmenter lip radius, distance from exhaust tube, ratio of augmenter diameter to exhaust tube diameter, etc. I’ve run several CFD tests based on the optimized augmenter design from previous research.
Figure 5-1: Running 50cm, valved, pulsejet with augmenter

Figure 5-2: Augmenter location as prescribed by Fei Zheng
5.1 Effect of Forward Flight Speed on Augmenter Performance

One area of research that has yet to be heavily investigated is the amount that the augmenter is affected by the forward flight speed. It is known that the augmenter increases thrust by increasing mass flux through its exit. The only way that this can work is by having the lip of the augmenter be located at an area greater than the outer diameter of the exhaust tube. This is because the pulsejet sends a vortex off of the flare, and this vortex grows and travels outward as it moves farther from the exit of the pulsejet [6]. By analysis done by Fei Zheng the optimal augmenter geometry calls for the augmenter radius to be greater that of the combustion chamber. Because the augmenter must have a greater cross sectional area than the jet itself there will be a drag penalty that increases with the square of the velocity when compared with the unaugmented jet. This means that with increased forward flight speed there must be a point where the drag of the augmenter actually outweighs the thrust increase, effectively rendering the augmenter useless. This also means that without changing

\begin{align*}
  d & \quad 1.25'' \\
  D & \quad 4.34'' \\
  r & \quad 0.75'' \\
  L & \quad 10'' \\
  X & \quad 2.75'' \\
  \delta & \quad 2'' \\
  \alpha & \quad 4.64^\circ
\end{align*}
the pulsejet design the best thrust augmentation possible is only achievable in static testing because there will be no aerodynamic drag.

Experimental tests have been conducted on a 50 cm valved pulsejet and computational simulations closely match the experimental results. I ran simulations on this 50 cm jet without the augmenter to get the baseline for the thrust produced. These tests were run with the premixed propane air valve as described in chapter 4 with a convective free stream range of 0 m/s to 40 m/s. The unaugmented valved jet produced approximately 19.5 N of thrust independent of the flow speed. I then re ran these same tests with an augmenter attached to find the thrust augmentation. A picture of this geometry is shown below in Figures 5-3 and 5-4.

*Figure 5-3: 50 cm valved pulsejet with augmenter half-section view*
5.1.1 Steady State Aerodynamic Drag

A CAD model of the optimal pulsejet with augmenter design was created for CFD tests. The first thing that was tested was how the augmenter itself responded to increasing forward flight speeds. I created a set of CFD tests on the augmenter by itself in a wind tunnel to determine how the drag increases with velocity. This is a very well understood aerodynamic force and these tests simply serve to gain a better understanding of the overall forces being applied on the augmenter. It is expected that the combustion and associated fluid dynamics of the running pulsejet will have a significant affect on the drag of the augmenter as well but it is important to understand how much drag there will be at minimum due to the aerodynamics while in the convective flow stream. The results of these drag simulations are
seen below in Table 5-1. From these results it is clear to see that, at low speeds, the augmenter generates negligible drag when compared with the 19.5 N of thrust created by the unaugmented valved jet. However, at speeds above 40 m/s the augmenter generates drag on the same order of magnitude of the unaugmented thrust. At first inspection this could be where the augmenter benefits start seeing incredibly diminishing returns.

Table 5-1: Drag on augmenter alone with increasing free stream velocity

<table>
<thead>
<tr>
<th>Free Stream Velocity (m/s)</th>
<th>Drag (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.00</td>
</tr>
<tr>
<td>15</td>
<td>0.72</td>
</tr>
<tr>
<td>25</td>
<td>1.95</td>
</tr>
<tr>
<td>40</td>
<td>4.93</td>
</tr>
<tr>
<td>50</td>
<td>7.66</td>
</tr>
<tr>
<td>60</td>
<td>10.97</td>
</tr>
</tbody>
</table>

5.1.2 Augmented 50 cm Pulsejet Thrust in Varying Convective Streams

Now that the aerodynamic drag of the augmenter has been calculated it is necessary to see what augmentation is actually generated. Here the amount of “extra” thrust generated by the augmenter will be called the augmentation ration and is represented by:

\[
\text{Augmentation Ratio} = \frac{\text{Total Augmented Thrust}}{\text{Thrust of Jet Alone}}
\]
This augmentation ratio will be used to compare how much thrust is being generated by the augmenter. Because the valved pulsejet by itself doesn’t produce any extra thrust with increased flight speed this augmentation ratio precisely gives a metric for how the augmenter increases thrust.

As seen in Table 5-2 there is significant thrust augmentation at static and low flight speeds. As the flight speed increases however, the thrust augmentation begins to drop monotonically. This should be expected due to the fact that the only reason that the thrust augmentation should drop is the increase in drag due to increased velocity.

<table>
<thead>
<tr>
<th>Free Stream Velocity (m/s)</th>
<th>Thrust of Jet Alone (N)</th>
<th>Thrust of Augmenter (N)</th>
<th>Total Thrust (N)</th>
<th>Augmentation Ratio</th>
<th>Isp (s)</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>19.54</td>
<td>22.61</td>
<td>42.15</td>
<td>2.16</td>
<td>4774</td>
<td>245</td>
</tr>
<tr>
<td>15</td>
<td>18.83</td>
<td>11.98</td>
<td>30.81</td>
<td>1.64</td>
<td>3204</td>
<td>221</td>
</tr>
<tr>
<td>25</td>
<td>19.14</td>
<td>5.49</td>
<td>24.63</td>
<td>1.29</td>
<td>2632</td>
<td>222</td>
</tr>
<tr>
<td>40</td>
<td>19.54</td>
<td>-5.33</td>
<td>14.21</td>
<td>0.73</td>
<td>1661</td>
<td>223</td>
</tr>
</tbody>
</table>

The -5.33 Newtons of thrust represents that the augmenter is actually hurting the performance of the jet with 5.33 N of extra drag. This general trend of thrust augmentation is shown below in Figures 5-5.
Where the augmented jet thrust line crosses the unaugmented jet thrust line is where the augmentation ratio is equal to 1. In this case that velocity is interpolated to 32 m/s. This is the velocity where the drag of the augmenter equals the extra thrust obtained from the augmentation. Running the pulsejet above this velocity results in a net loss in thrust. These results are somewhat unprecedented and show that for the 50 cm valved pulsejet, which is a common design amongst hobbyists, it isn’t feasible to run the pulsejet with an augmenter above 32 m/s corresponding to about 72 miles per hour.

Another point of interest to note on these tests involves the measurement of the unaugmented thrust of the pulsejet. In the experimental case the augmentation ratio is
determined by running the jet unaugmented and recording its thrust, then running the jet with the augmenter and determining its thrust, and dividing those two numbers. This is how the augmentation ratio was calculated above. CFX has the capability of measuring the thrust on the unaugmented jet while the augmenter is actually present. This presents a unique look at how the augmenter interacts with the jet while it is running, something not possible in the experimental case. When looking at the thrust measured at the exit of the exhaust tube on a running pulsejet with augmenter it shows that, statically, the jet has an increase in thrust. This thrust actually decreases as the forward flight speed increases and then falls below the jet alone thrust at the same point that the augmentation ratio becomes less than 1. This is surprising because the jet without an augmenter does not see any appreciable response in thrust to the increase in forward flight speed. Table 5-3 shows the thrust measured at the exit of the pulsejet but with an augmenter behind it.

Table 5-3: Thrust measured at the exit of the pulsejet with augmenter

<table>
<thead>
<tr>
<th>Free Stream Velocity (m/s)</th>
<th>Thrust (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>25.77</td>
</tr>
<tr>
<td>15</td>
<td>21.11</td>
</tr>
<tr>
<td>25</td>
<td>19.8</td>
</tr>
<tr>
<td>40</td>
<td>16.44</td>
</tr>
</tbody>
</table>

This is a result that has never been seen because it hasn’t previously been possible to measure experimentally. I postulate that this is due to the fact that the pulsejet operates subsonically by nature and thus there will be some communication between the front of the augmenter and
the exhaust tube of the pulsejet. What is unknown yet is how the augmenter actually causes this.
CHAPTER 6

Further Experiments

In addition to all of the research presented above in this paper, I have conducted a multitude of other tests to help with the design of further experimental pulsejets. These tests are useful for supplemental material and further research in the field. All of these CFD tests were designed with future work in mind.

6.1 Wind Tunnel Testing

6.1.1 Experimental Wind Tunnel

All of the experimental research done at the AERL at NCSU has been done statically, meaning there is no forward flight speed. While that is very beneficial for studying the operating nature of the pulsejet it isn’t very practical for designing something that one hopes to actually use as a means of propulsion. In the fall of 2009 we built a physical wind tunnel to help get some dynamic testing done. Figure 6-1 shows this tunnel. The wind tunnel was built with a 6” inner diameter and is capable accelerating air between 10 m/s and 30 m/s. Preliminary tests showed that on a typical fall day the accelerated air had a very constant velocity profile throughout the test section within that 20 m/s range and had an average flow
temperature of 88° F. These flow properties were used in my CFD and compared with the experimental data to test the fidelity of the computational simulations.

Figure 6-1: Wind tunnel with pulsejet on sting and dynamometer

A dynamometer was used to obtain force measurements from the pulsejet. The pulsejet is mounted on a sting to transmit the forces down through the bottom of the wind tunnel and into the dynamometer. For all of the drag testing a NACA 0012 airfoil was used as a sting because it is symmetric and, if lined up with zero angle of attack, will produce no aerodynamic lift. A close up of the dynamometer is shown in Figure 6-2, where the
extremely thin ribs allow for the transmission of any forces on the pulsejet to the dynamometer’s sensors.

![Image of dynamometer](image)

Figure 6-2: Close up on dynamometer

### 6.1.2 CFD Wind Tunnel

I created an identical wind tunnel in ANSYS and ran several drag tests on the 12 centimeter pulsejet shown above. Figure 6-3 shows the CAD drawing for the pulsejet in the 6” tunnel with the NACA 0012 sting. The setup for this test was very simple because there was no chemistry going on in the actual pulsejet. The runs were steady state and set in an atmosphere equivalent to the tests done experimentally. A series of runs were done with free stream velocities ranging from 20 m/s to 30 m/s. These tests were done with the jet on the sting, as in the experimental model, as well as with the sting alone to subtract the drag from
the combination tests. Table 6-1 shows the results from these tests as well as the calculated coefficient of drag.

*Figure 6-3: CFD 6” wind tunnel with 12 cm pulsejet on NACA 0012 sting*

The Coefficient of drag was calculated from the following formula, the drag equation [10]:

\[ C_d = \frac{D}{\frac{1}{2} \rho v^2 A} \]
**Table 6-1: Drag and \( Cd \) for 12 cm jet on NACA 0012 sting in 6” CFD wind tunnel**

<table>
<thead>
<tr>
<th>Velocity (m/s)</th>
<th>Drag Jet and Sting (mN)</th>
<th>Drag Sting (mN)</th>
<th>Drag Jet (mN)</th>
<th>( Cd )</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>59.67</td>
<td>31.43</td>
<td>28.24</td>
<td>0.277</td>
</tr>
<tr>
<td>21</td>
<td>65.41</td>
<td>34.39</td>
<td>31.01</td>
<td>0.276</td>
</tr>
<tr>
<td>22</td>
<td>71.38</td>
<td>37.47</td>
<td>33.91</td>
<td>0.275</td>
</tr>
<tr>
<td>23</td>
<td>77.60</td>
<td>40.67</td>
<td>36.94</td>
<td>0.274</td>
</tr>
<tr>
<td>24</td>
<td>84.09</td>
<td>43.98</td>
<td>40.11</td>
<td>0.273</td>
</tr>
<tr>
<td>25</td>
<td>90.80</td>
<td>47.40</td>
<td>43.40</td>
<td>0.273</td>
</tr>
<tr>
<td>26</td>
<td>97.75</td>
<td>50.94</td>
<td>46.80</td>
<td>0.272</td>
</tr>
<tr>
<td>27</td>
<td>104.95</td>
<td>54.60</td>
<td>50.35</td>
<td>0.271</td>
</tr>
<tr>
<td>28</td>
<td>112.37</td>
<td>58.36</td>
<td>54.00</td>
<td>0.270</td>
</tr>
<tr>
<td>29</td>
<td>120.03</td>
<td>62.24</td>
<td>57.79</td>
<td>0.270</td>
</tr>
<tr>
<td>30</td>
<td>127.94</td>
<td>66.23</td>
<td>61.71</td>
<td>0.269</td>
</tr>
</tbody>
</table>

It is interesting to note that there is a 3% drop in the coefficient of drag as the tests increase from 20 m/s to 30 m/s. I attribute this to the fact that there are effects other than simple steady state flow going on. The pulsejet is not a bluff body and has an opening through the centerline. The entire jet is acoustically resonant and varying the flow speed definitely changes the static pressure as well as the flow characteristics inside of the jet. This could contribute to the decreasing coefficient of drag on the jet. These numbers were
compared with the experimental data obtained from running the same jet in the wind tunnel and the plot is presented in Figure 6-4.

![Figure 6-4: Drag versus free stream velocity, computational drag compared with experimental drag](image)

When the computational results are compared with the experimental data it is easy to see that there is strong predictability of the computer model. The CFD results show a high correlation and the drag for both cases is nonlinear, as predicted by the drag equation. It is important to note that the experimental measurements were made on two different days, thus the assumption of 88° F and exactly 1 atmosphere of pressure will be slightly off.
Nevertheless the computer model predicts, within a few percent, the drag in the overlapping areas. Because of this high correlation I did not do anymore CFD on the lower flow regimes.

### 6.2 Cold Flow Testing

To gain a better understanding of valve dynamics an experimental testing rig is being built to allow for viewing and recording of a valve in motion. The problem with viewing a valve in motion on a running pulsejet is that the pulsejet head normally obstructs the view of a valve often making it difficult to even see the valve in motion. Also, current pulsejets have proven to be quite difficult in reproducing consistent tests. For the purpose of studying the valve motion we want to be able to test in a controlled environment with high reproducibility.

To create this controlled environment Joe Scroggins designed a cold flow testing chamber which houses a volume the size of a normal pulsejet combustion chamber where air is pulsed through. This testing rig is designed to pull the average chamber pressure down to -7 psi while the pulsed air brings the pressure up to 14 psi. This range was decided upon as the normal operating range of a pulsejet. The chamber has threading for attaching interchangeable pulsejet heads without an inlet. This allows the viewer an unobstructed view of the valves as they move. There is no fuel injection and thus no combustion. This “cold flow” test is solely for studying how the valves behave under the pressures involved. A picture of an early iteration of the cold flow testing rig is shown here in Figure 6-5.
The cold flow testing rig is consistently going through design iterations in order to nail down the best design to produce the desired chamber pressures. Due to lack of current experimental tests on the rig I am presenting CFD of the rig to confirm feasibility of the device as well as an analysis of how it works.

6.2.1 Cold Flow CFD Testing

A simple cold flow device based on the design specs of the experimental rig is shown in Figure 6-6. The simulations were done in 2-D as a 4° wedge, as in the pulsejet tests. The computational model consists of a high pressure chamber stored at a known pressure, a
middle chamber section, and a low pressure chamber held constant at -7 psi. The inlet between the high pressure chamber and the middle chamber is a 5/8” opening. Between the middle chamber and the -7 psi low pressure chamber is a 1/2” opening.

The experimental testing rig aims to reproduce valve testing for the 12 cm pulsejet up to the 50 cm pulsejet. Therefore, the rig must be able to produce the -7 psi to 14 psi range with a response time between 250 Hz and 1500 Hz. Steady state testing was conducted and showed that a high pressure chamber pressure of 123 psi was sufficient to pull from -7 psi up to the desired 14 psi so I began the transient testing there. This pressure was more than sufficient to respond at 250 Hz of pulsed high pressure air so I moved to 1000 Hz test. At high pressure chamber at 123 psi the rig was not able to reach the desired 14 psi peak. Testing was continued with high pressure chamber pressures at 133 psi, 142 psi, and 155 psi.

*Figure 6-6: CFD cold flow testing device*
and only the 142 psi and 155 psi tests were able to achieve the 14 psi peak. Figure 6-7 shows the testing chamber pressure for the 123 psi, 133 psi, and 142 psi tests.

![Graph showing testing chamber pressure for various high pressures](image)

Figure 6-7: Cold flow testing at 1000 Hz for various high pressures

Surprisingly the low pressure actually pulls the chamber down to -10 psi where in initial testing it was only capable of pulling down to -7 psi. This isn’t a problem however as we are only concerned with the ability of the testing rig to pull a 21 psi range. When this is considered it is easy to see that the high pressure chamber of 123 psi doesn’t come close to reaching this goal. The high pressure chamber of 133 psi comes very close and the high pressure chamber of 142 psi easily achieves this goal. Because of these outputs I did not test the 155 psi case at 1000 Hz. I did test 155 psi at 1500 Hz as well as at 2000 Hz. The high pressure chamber of 155 psi easily reached the 14 psi test chamber goal at 1500 Hz. At 2000
Hz the 155 psi test showed that it was able to reproduce an average test chamber of 14 psi but at individual points throughout the chamber there were spots well below 14 psi and others slightly above. This leads to the conclusion that the fluid mechanics of the device were not responding fast enough for the necessary pressure changes to occur at 2000 Hz. This isn’t a problem however because the fastest that we would want to test with this current design is for the 12 cm pulsejet head which operates in the 1000 Hz to 1500 Hz range.

These tests show that it is feasible to have this design reproduce the necessary response to drive the valve motion at 1000 Hz. From this I was able to extract the average mass flux from the high pressure side. Because these pressures and temperatures fall well within the assumptions for the Ideal Gas Law to be valid I was able to calculate the necessary high pressure chamber volume [11]. This data is shown in Table 6-2.

Table 6-2: Average mass flux and necessary high pressure chamber volume at 1000 Hz

<table>
<thead>
<tr>
<th>High Pressure Chamber Pressure (psi)</th>
<th>Average Mass Flux (g/s)</th>
<th>Necessary High Pressure Chamber Volume (m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>123</td>
<td>13.78</td>
<td>1.021</td>
</tr>
<tr>
<td>133</td>
<td>19.09</td>
<td>1.420</td>
</tr>
<tr>
<td>142</td>
<td>25.50</td>
<td>1.580</td>
</tr>
<tr>
<td>155</td>
<td>31.80</td>
<td>1.741</td>
</tr>
</tbody>
</table>

These cold flow tests are based on keeping a constant high pressure chamber pressure. While that isn’t very practical in an experimental sense, we can approximate a constant high pressure chamber if we build a large enough high pressure chamber. Using the average mass flux I was able to calculate the total mass that leaves the high pressure chamber
during a run. We know that the pressure in the high pressure chamber will decrease once mass flows out so it is important to know how big of a decrease actually occurs during a run [11]. Assuming that a 30 second run time is adequate to capture data and assuming that the Ideal Gas Law is still valid I was able to calculate the necessary high pressure chamber volume to only have a 5% loss in pressure at the end of the run. This would still allow the high pressure chamber of 142 psi and 155 psi to still be able to produce the necessary pressure ranges at 1000 Hz. These necessary high pressure chamber volumes are also presented in Table 6-2.
CHAPTER 7

Conclusions

The work on valveless, unaugmented and augmented valved, and curved pulsejets presents an interesting view on how these pulsejets run in varying forward flight speeds. This also leads to future work that can be done to experimentally test these jets based on the CFD presented. These are several conclusions that can be drawn from the work presented above:

• The valveless pulsejet is very inefficient when compared with a valved or curved pulsejet. They should only be used if it is the only option for running a pulsejet.
• U-shaped pulsejets indeed regain nearly all of the lost thrust due to expelling combustion products out of the inlet of the jet.
• The valved pulsejet is provides increased thrust over the valveless jet and adding an augmenter can provide additional thrust as well.
• The augmenter is useful to increase thrust only if it is below 32 m/s at which point the drag penalty is too large.
• Modeling the running pulsejet in a wind tunnel on a sting is very accurate when compared with the experimental wind tunnel data
• CFD tests show feasibility of building a cold flow chamber that can respond fast enough to emulate valve motion without the need of running a jet with combustion
REFERENCES


Appendix A: FORTRAN code for valve dynamics

#include "cfx5ext.h"

dllexport(mdot1)

    SUBROUTINE Mdot1 ( 
        & NLOC, NRET, NARG, RET, ARGS, CRESLT, CZ,DZ,IZ,LZ,RZ )

    CC
    CD Uses value from a previous iteration in a calculation
    CC

    CC --------------------
    CC      Input
    CC --------------------
    CC
    CC NLOC    - size of current locale
    CC NRET    - number of components in result
    CC NARG    - number of arguments in call
    CC ARGS()  - (NLOC,NARG) argument values
    CC

    CC --------------------
    CC    Modified
    CC --------------------
CC
CC  Stacks possibly.
CC
CC -----------------------
CC       Output
CC -----------------------
CC
CC  RET() - (NLOC,NRET) return values
CC  CRESLT - 'GOOD' for success
CC
CC -----------------------
CC       Details
CC -----------------------
CC
CC
CC============================================
C
C
C -----------------------
C  Preprocessor includes
C -----------------------
C
#include "parallel_partitioning.h"
C
C -----------------------------
C    Global Parameters
C -----------------------------
C
C
C
C -----------------------------
C    Argument list
C -----------------------------
C
C    INTEGER NLOC,NARG,NRET
C
C    CHARACTER CRESLT*(*)
C
C    REAL ARGS(NLOC,NARG), RET(NLOC,NRET)
C
C    INTEGER IZ(*)
C    CHARACTER CZ(*)(1)
C    DOUBLE PRECISION DZ(*)
C    LOGICAL LZ(*)
C    REAL RZ(*)
C
C
C -----------------------------

67
C External routines
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C Local Parameters
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------

CHARACTER*20 ROUTIN

PARAMETER(ROUTIN='HEAT_SOURCE')
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------
C                             
C                             
C                             
C-------------------------------

REAL PCC, PUP, MDOT, VV, XV, DT, AIN, LIN, AV, MV, PIN

INTEGER STEP
C                             
C                             
C                             
C initialize air conditioner to be off

DATA PUP/0.0/, MDOT/0.0/, VV/0.0/, XV/0.0/, T/0.0/, PIN/0.0/

SAVE PUP, MDOT, VV, XV, T, PIN
DATA AIN/3.7476e-4/, LIN/0.0508/, G/1./, RHO/1.181/, AVX/0.0693/
DATA MV/0.00227/, AVMIN/3.7476e-6/, KV/200/, DT/1E-5/

C

C ------------------------------

C Stack pointers

C ------------------------------

C

C==============================================================================================================

C

C ------------------------------

C Executable Statements

C ------------------------------

C

C

C Initialise RET() to zero.

CALL SET_A_0( RET, NLOC*NRET )

C

IF (ARGS(1,2) .NE. T) THEN

   DT=ARGS(1,2)-T

   IF (DT .GT. 1E-5) THEN

      DT = 1E-7

   ENDIF

ENDIF
ENDIF

T=ARGS(1,2)

PCC=ARGS(1,1)

PIN=ARGS(1,3)

VV=VV+1/MV*(AIN*(PUP-PCC)-KV*XV)*DT

XV=XV+VV*DT

IF (XV .LT. 0.0) THEN
    XV=0.0
    VV=0.0
    MDOT=0.0
    PUP=0.0
ELSE

    AV=MAX(AVMIN, AVX*XV)

    DO I=1,10

        IF (MDOT .LT. 0) THEN
            PUP=PCC-MDOT**2/(2*RHO*AV**2)
        ELSE

            ENDIF
PUP = PCC + MDOT**2/(2*RHO)*(1/AV**2-1/AIN**2)
ENDIF

MDOT = MDOT + AIN/LIN*(PIN-PUP)*DT/10.0

IF (MDOT .GT. 0) THEN
    PIN = -MDOT**2/(2*RHO*AIN**2)
ELSE
    PIN = 0.0
    MDOT = 0.0
    PUP = 0.0
    XV = 0.0
    VV = 0.0
ENDIF
ENDDO
ENDDO
ENDIF
ENDIF
ENDIF

RET(1,1) = MDOT

C Set success flag.
CRESLT = 'GOOD'

C

C=====================================================

END