141
220
THS

2008

LIBRARY
Michigan State
University

This is to certify that the
thesis entitled

DESIGN, FABRICATION AND TEST OF A GAS TURBINE
ENGINE AND WAVE ROTOR TEST BED

presented by

JOHN CHARLES QUACKENBUSH, II

has been accepted towards fulfillment
of the requirements for the

M.S. degree in MECHANICAL ENGINEERING

N. Mu

Major Professor's Signature

8/13/08

Date

MSU is an affirmative-action, equal-opportunity employer

PLACE IN RETURN BOX to remove this checkout from your record.
TO AVOID FINES return on or before date due.
MAY BE RECALLED with earlier due date if requested.

DATE DUE	DATE DUE	DATE DUE

5/08 K:/Proj/Acc&Pres/CIRC/DateDue.indd

DESIGN, FABRICATION AND TEST OF A GAS TURBINE ENGINE AND WAVE ROTOR TEST BED

By

John Charles Quackenbush, II

A THESIS

Submitted to
Michigan State University
in partial fulfillment of the requirements
for the degree of

MASTER OF SCIENCE

Mechanical Engineering

2008

ABSTRACT

DESIGN, FABRICATION AND TEST OF A GAS TURBINE ENGINE AND WAVE ROTOR TEST BED

By

John Charles Quackenbush, II

This thesis examines two core technologies developing within the energy production and transfer discipline: smaller scaled gas turbine engines and performance enhancing wave rotors. A combustion chamber was designed to satisfy the requirements to operate both devices. The chamber was built to be integral within the test apparatus, which was outfitted with a precise data acquisition system to critically monitor and evaluate these technologies in real-time.

Both the wave rotor and gas turbine engine prototype were tested. The testing was done separately to first fully understand these technologies in order to later enhance the gas turbine engine with the integration of the wave rotor as an engine topping performance enhancer.

This thesis will discuss the analysis and decisions made to design and fabricate this test apparatus. Data were collected for both devices allowing for solid evaluation and recommendations on how to further proceed.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURE

A	Area
D	Diameter or width
I.T.T.	Inlet turbine temperature
m	Mass flow rate
P	Pressure
R	Gas constant for air (287 kJ/Kg-K)
T	Temperature
V	Velocity
π	Constant (3.1416)
ρ	Density
$\mathbb{V}$	Volumetric flow rate

INTRODUCTION

This thesis is an advancement of previous work as well as a venture into newer realms of research. The goals of this thesis are to examine two core technologies developing within the turbomachinery discipline: smaller scaled gas turbine engines and performance enhancing wave rotor devices.

In previous work a complex Brayton cycle analysis was conducted for a specific turbine–compressor hardware couple for the purpose of developing a common gas turbine engine capable of 5 kW electrical power generation [Kusner 2006]. This thesis expands upon this work with the design of an appropriate combustion chamber to drive the turbine-compressor couple to produce a desired power output. Once the combustor was complete the engine was evaluated by direct testing within a precisely monitored test apparatus utilizing a real-time data acquisition system.

This project was also to simultaneously develop and test a specific wave rotor technology. As later discussed, the wave rotor is a non-steady pressure exchange device that can operate within a common engine cycle. Its purpose is to increase the pressure of the main engine air intake before combustion. As a result the wave rotor has the ability to increase efficiency and enhance the overall performance of many engine types: gas turbine engine or others.

Like the gas turbine engine efforts discussed above the wave rotor technology will be evaluated through direct testing. A test rig analogous to the gas turbine engine rig must be developed. To properly evaluate the wave rotor, the high temperature points that it would naturally operate in as a typical engine cycle enhancer must be achieved. It was determined that a parallel design path should be taken. The combustor that is needed for the gas turbine engine should also be designed to be capable of generating the operating conditions needed to evaluate the wave rotor at real operating points. Like the combustor, the data acquisition system will also be designed to satisfy the needs of the gas turbine engine as well as the wave rotor testing.

Figure 1 shows a flow chart of what has already been accomplished with the gas turbine engine and wave rotor developments as well as the design, building, and testing objectives of this thesis. This figure concludes with the pinnacle goal of these efforts after rigorous analysis and testing being the coupling of the two technologies. The gas turbine engine will be enhanced with the wave rotor proving both technologies.

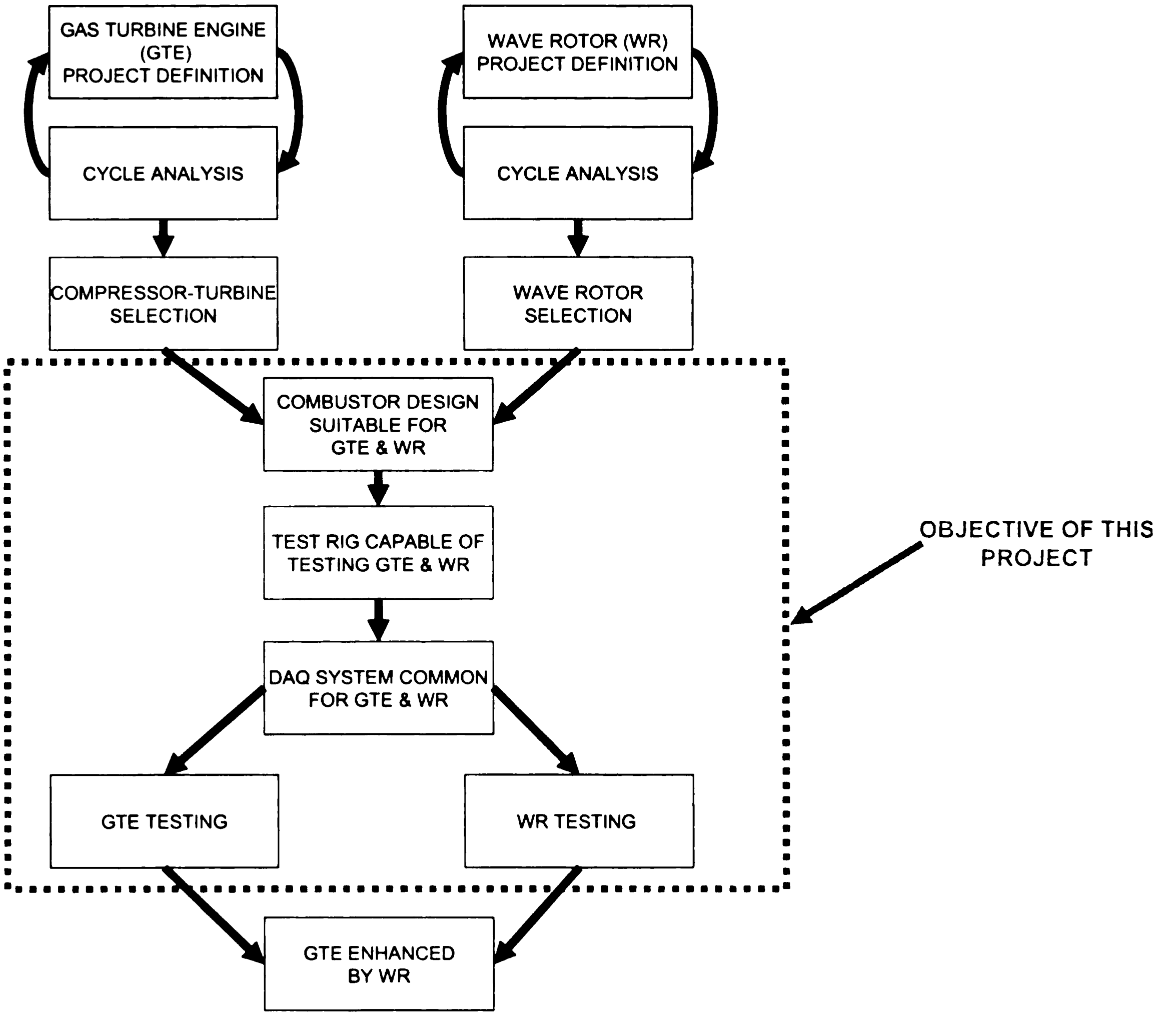

Figure 1. Project goals and direction

TEST HARDWARE

TURBOCHARGER

The design intent for the turbocharger technology is to couple it with an appropriate combustion chamber in a closed-loop orientation such that it can operate as a typical gas turbine engine. Gas turbine engines are common as off-the-grid electrical power sources, generally at larger power outputs. This engine aims to generate power at a smaller scale, 5 kW, which would prove to be more useful for personal residential uses. The niche in the electrical generator market is currently dominated by reciprocating internal combustion engines commonly running on gasoline or diesel fuel. This gas turbine engine will be run with propane as fuel.

A specific turbocharger and the precise Brayton cycle in which it needs to operate within were previously determined by the predecessor of this project [Kusner, 2006]. He determined to use a turbocharger from Garrett which is a subsidiary of Honeywell. Specifically the turbocharger is a Garrett GT1241 model, which is the smallest in the line that Garrett offers. This turbocharger, when used in automotive applications is recommend for internal combustion engines with 0.5 to 1.2 liters of displacement and capable of 50 to 130 horsepower [Garrett, 2008].

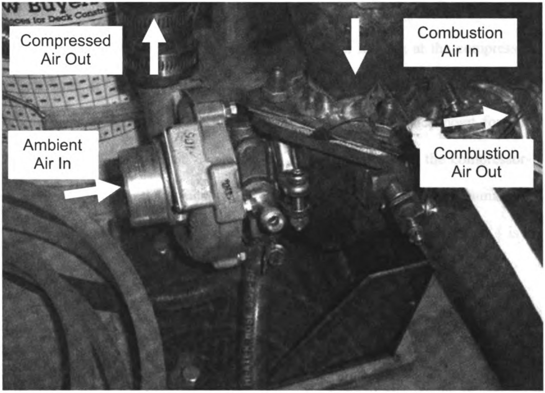

Figure 2. Garrett GT1241 turbocharger, basic operating principle

The Brayton cycle analysis previously executed has already set the operating point needed to run the gas turbine engine to produce the goal of 5 kW of power. The operating point has been determined to be 0.071 kg/s (roughly 9.4 lbs/ min) of mass flow rate through the system. The turbocharger will ingest air at standard atmospheric pressure and temperature. The pressure ratios of the compressor and turbine have both been set to be 2.5 [Kusner, 2006]. This means that the 101.3 kPa atmospheric air entering the compressor will be increased to 253.3 kPa. After being ignited in the combustion chamber, this hot gas will expand across the turbine and be brought back to near atmospheric pressure before being exhausted (Figure 2). In order for the compressor and turbine to match in pressure and flow there must be an energy input to the system to overcome the mechanical losses. This energy will be added in the form of heat addition to the air after it has been compressed but before entering the turbine. By the cycle analysis, this heat addition has been calculated so that the 0.071 kg/s of air flow must be elevated to 1200 Kelvin before entering the turbine. When looking at the compressor map for the Garrett GT1241 (Figure 3), it can be seen that the operating point of 9.4 lbs/min and a pressure ratio of 2.5 falls nicely at the highest efficiency point for this chosen turbocharger, 76% efficient. Note that at this operating point the compressor-turbine couple will be spinning at roughly 215,000 RPM. Table 1 shows a summary of the operating condition that must be designed and tested [Kusner, 2006]. Figure 4 is a turbine map analogous to the compressor map.

Table 1. Gas turbine engine 5 kW operating point

Turbocharger - GTE Operating Point		
Mass Flow	Combustor Temperature	Combustor Pressure
0.071 kg/s	1200 K	253 kPa

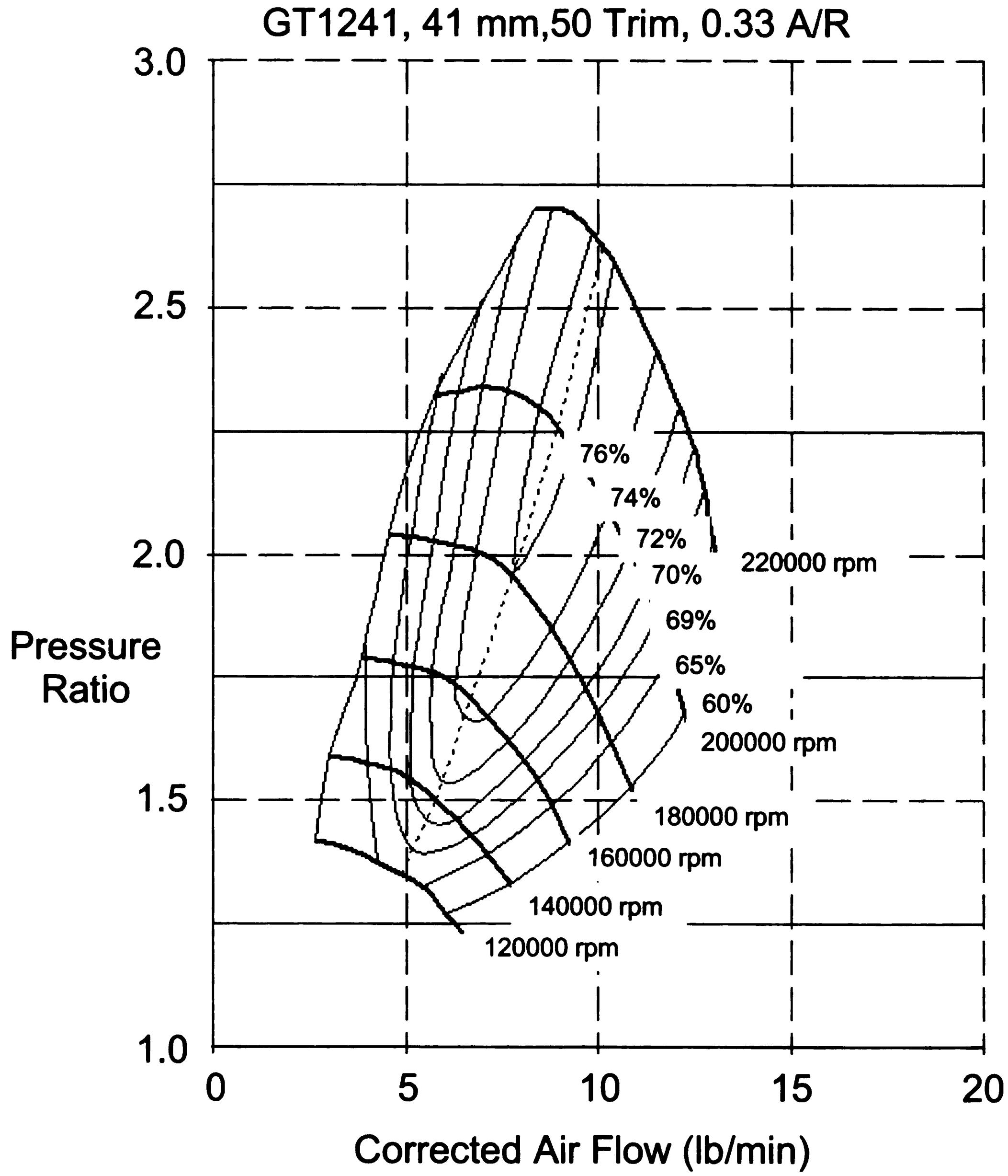

Figure 3. Compressor map, Garrett GT1241 manufacturer data [Garrett, 2008]

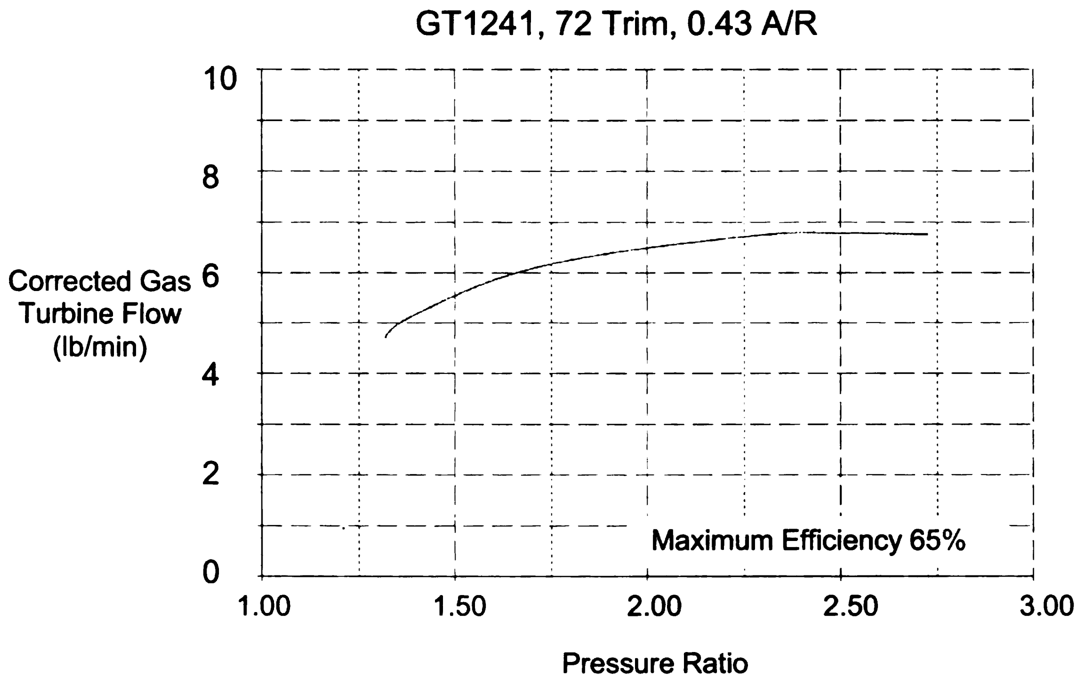

Figure 4. Turbine map, Garrett GT1241 manufacturer data [Garrett, 2008]

WAVE ROTOR

A wave rotor uses shock waves to pressurize fluids by transferring energy from a high-pressure flow to a low-pressure flow. It consists of many curved or straight channels arranged around the axis of a rotating drum or disc.

Specifically for this project, an automotive Comprex 7 type waver rotor was used for testing. Roughly 150,000 of Mazda's 626 powered by a diesel engine were supercharged with a Comprex pressure wave supercharger [Dempsey, 2006]. Mazda used this particular wave rotor design as a performance enhancement to this automotive internal combustion engine. This project intends to build a test rig capable of testing this wave rotor at operating points that would simulate its application as a gas turbine enhancer.

A wave rotor works with the propagation of shock and expansion waves. Figure 5 is a schematic view of how a wave rotor could be connected with a typical gas turbine engine. The incoming air is first compressed by the engine's main compression stages (state 0 to 1). This compressed air then enters the wave rotor channels through specifically designed porting. Within these channels energy transfer takes place. The cool, compressed air is exposed to hot combustion gases from the combustion chamber (state 3). The pressure and temperature difference between the two fluids induces a shock wave that is used to compress the cool air. The pressurized air is fed to the combustion chamber (stage 2). On the other hand, the hot exhaust gases are expanded, leave the wave rotor (stage 4) and are further expanded across the engines turbine stages [Akbari, 2006].

The wave rotor spins at a specific speed about its central axis. This speed and the porting geometry that controls the flows in and out of the wave rotor are precisely determined such that the compression and expansion processes take place and enter and leave the rotor at the correct ports.

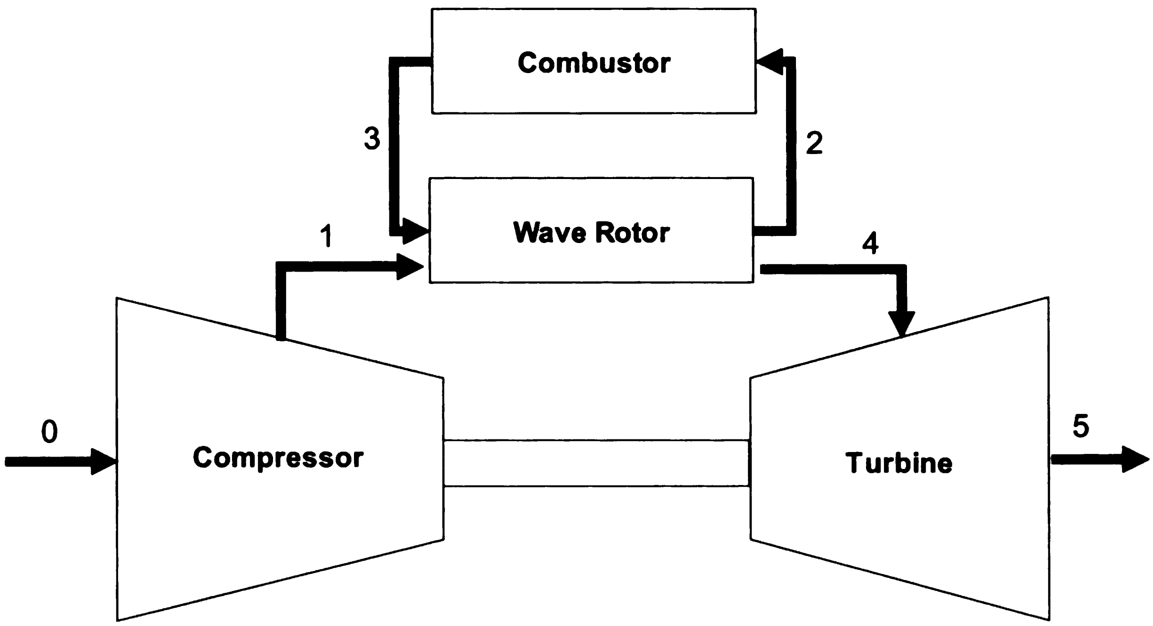

Figure 5. Wave rotor, schematic view showing flow directions

Figure 6 is a top view of this specific wave rotor. This wave rotor is of the straight channel type.

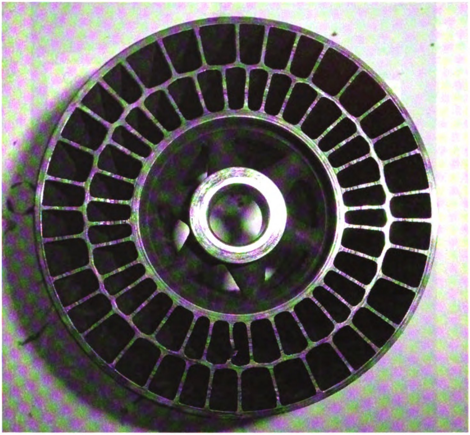

Figure 6. Straight channel wave rotor

The main objective for this test rig will be to evaluate operating conditions by directly simulating them. Table 2 shows the first operating point to be tested and validated to. This operating point serves as a design constraint on the test rig, meaning that the rig must be capable of this operating condition. It is worth noting that the wave rotor will be spinning at 13,000 rpm at this operating point. The electric drive motor to accomplish this can be seen in Figure 7.

Table 2. Wave rotor first test and validation operating point

Wave Rotor Operating Point		
Mass Flow	Combustor Temperature	Combustor Pressure
0.056 kg/s	850 K	250 kPa

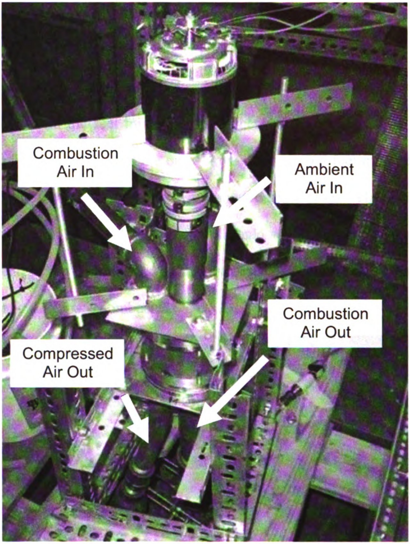

Figure 7. Wave rotor, basic operating principle

COMBUSTION CHAMBER DESIGN

PREVIOUS COMBUSTOR

The previous combustor was an inner annulus design [Kusner, 2006]. This annular design was made of an inner and outer pair if pipes both equally concentric about the flow direction. The outer pipe was the same 2-inch diameter, schedule 40, steel pipe of which the rest of the rig was made. The inner can (Figure 8) was made of a 1-inch diameter, schedule 40, steel pipe. A large majority of the flow was designed to pass around this inner pipe. This bypassing air is not needed for the combustion process and can serve as cooling air for the much hotter combustion inside. Some of the air enters the inner can of the combustor. Shortly downstream of where the air enters, the fuel is injected. The air and the fuel mix inside the inner can with the help of mixing holes drilled perpendicularly through to the outer flow. This air and fuel mixture will then be ignited at the end of the inner can by two automotive spark plugs. The spark plugs were placed at the end with the intent that the flame would attach itself to the end of the inner pipe. At the end of the inner can the air that bypassed will then be able to mix back with the hot exhaust gas that was combusted.

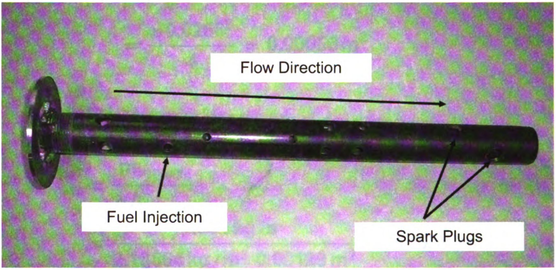

Figure 8. Previous combustor design, inner annulus type

This design was tested by Michael Kusner [Kusner, 2006]. His testing showed that there were issues maintaining stable combustion. Some combustion was witnessed, but it was only in short bursts that seemed to correlate with the spark ignitions of the automotive sparkplugs used. This intermittent ignition made it impossible for the test rig to reach any reasonable operating point. Besides the intermittent ignition the other assignable cause was determined to be the geometry of the combustion chamber. It is desired to run the test rig at a mass flow rate of .071 kg/s. It was previously determined that 0.030 kg/s of this mass flow is needed for combustion and the remaining mass flow bypasses the inner can [Kusner, 2006]. Using continuity, the bulk velocity of the air can be determined for the bypass and inner can flows.

$$A_{bypass} = \frac{\pi (D_1^2 - D_2^2)}{4}$$
$$= \frac{\pi (0.0525^2 - 0.0334^2)}{4} = 0.00129\,m^2$$

$$A_{combustion} = \frac{\pi D_3^2}{4}$$
$$= \frac{\pi\, 0.0266^2}{4} = 0.00056\,m^2$$

$$m = \rho A V$$

$$V_{bypass} = \frac{m_{bypass}}{\rho A_{bypass}}$$
$$= \frac{.041\ kg/s}{2.18864\ kg/m^3\ 0.00129\ m^2} = 14.52\ m/s$$

$$V_{combustion} = \frac{m_{combustion}}{\rho A_{combustion}}$$
$$= \frac{.030\ kg/s}{2.18864\ kg/m^3\ 0.00056\ m^2} = 24.48\ m/s \qquad (1)$$

These velocities are much too fast to sustain stable combustion of propane. The flame speed of propane at this operating point has been determined to be roughly 0.5 m/s

[Kusner, 2006]. As calculated, the velocities of the air through the combustor are much larger than the flame speed of propane, meaning that the air is capable of pushing the flame downstream faster than the flame can travel back upstream. This allows the flame to detach itself from the inner can, leave the combustor and travel downstream. This is called a flame out. High velocities are likely the cause of the flame propagation and explain why intermittent ignition was witnessed during testing. It was determined that a new combustion chamber design was needed. Any new design needs to take these high speed air velocities into account to prevent and minimize the likelihood of flaming out the combustor.

DUMP COMBUSTOR

The flame speed of propane has previously been calculated to be around 0.5 m/s [Kusner, 2006]. In order to have a stable flame it is important that any combustion chamber design have some region where the air is decelerated to this velocity. This decelerated region would be where the flame could stabilize and support sustainable combustion. The first design alternative that was examined to accomplish this deceleration was a simple dump combustor design. The advantage of a dump combustor is that it is capable of quickly expanding and decelerating the incoming air to a much larger diameter over a short axial length. Unfortunately, this sudden expansion causes a large pressure drop in the combustor. To evaluate this as a possible option, a specific geometry was analyzed using Fluent (Figure 9), a computational fluid dynamics software.

The air will be entering the combustor through a standard 2-inch-diameter inlet which corresponds to the diameter the air is extracted from the compressor. Using the continuity principle it was found that roughly a 12.77-inch-diameter cross section was needed to decelerate the air to a velocity below 0.5 m/s at the desired mass flow rate of 0.071 kg/s.

$$
\begin{aligned}
&\mathbf{m} = \rho \mathbf{A V} \\
&\mathbf{m} = \mathbf{0.071\ kg/s} \\
&\rho = \mathbf{2.189\ kg/m^3} \\
&\mathbf{V_{air}} = \mathbf{V_{flamespeed}} = \mathbf{0.5\ m/s} \\
&\mathbf{A} = \frac{\mathbf{m}}{\rho\, \mathbf{V_{flamespeed}}} = \mathbf{0.0649\ m^2} = \mathbf{100.60\ in^2} \\
&\mathbf{D} = \frac{\mathbf{4\sqrt{A}}}{\pi} = \mathbf{12.77\ in}
\end{aligned}
\tag{2}
$$

This quick calculation essentially determined the overall geometry of the dump combustor. The axial length of the combustor was limited to 24 inches in length for practical purposes and also to be easily adapted in to the current space available. The analysis showed that at the desired mass flow rate the velocity of the air in the 2-inch-diameter inlet is roughly 17 m/s which is well above the desired 0.5 kg/s target. The sudden expansion to the 12-inch-diameter does cause the air to decelerate. Near the outer walls of the large diameter region the air decelerates down to the desired velocity. The large majority of the air remains at high velocity and quickly passes and exits through the center of the combustor. This design would be undesirable. Most of the air would not be able to support the combustion process due to these high velocities. The flame front would be limited to the outer region of the combustor where there is not enough air to mix with the propane to provide the heat input that is needed to reach the operating point.

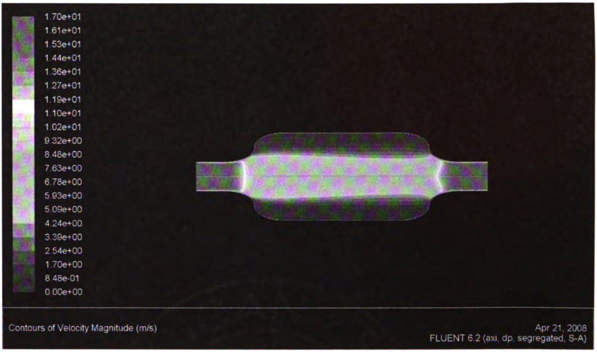

Figure 9. Dump combustor, velocity contour

INNER CONE COMBUSTOR

An initial design was to use an inner cone as a type of baffle to create low velocity regions in the flow that could provide stabilization for the flame. The main advantage to having an inner cone baffle design would be its simplicity. The outer diameter can remain consistent with the rest of the 2-inch-diameter piping. One draw back is that the flow must be decelerated to reach the desired velocities by introducing a pressure loss caused by the baffles. Again a representative Fluent model was executed. As before, the velocity of the air entering the combustor is roughly 17 m/s. Figure 10 indicates some low speed regions were developed down stream of the obstructions placed in the flow. These same obstructions also cause several areas where the flow was accelerated because it was being forced to flow around the baffles. Also it was found that at a 2-inch-diameter cross section it takes a considerable amount of blockage to create a slow 0.5 m/s region. This blockage causes an unacceptable amount of pressure loss. The analysis showed that only 9 percent of the total pressure at the inlet is maintained at the outlet. Therefore, the inner cone design was deemed not to be the best option. The air must be expanded and decelerated in a way that conserves pressure yet still reaches the desired velocities.

Figure 10. Inner cone combustor, velocity contour

COMBUSTION BYPASS

Due to previous analysis it was determined that the air must be decelerated by diffusion rather than a pressure loss device. This way a larger portion of the high velocity air can be transferred to static pressure rather than simply lost. Due to the large mass flow rate it would take a cross section equivalent to a 12.77-inch diameter to decelerate the entire flow of air to the 0.5 m/s flame speed of the fuel. The combustion process only needs a flow rate of 0.03 kg/s to reach the energy input for the desired operation point. This means that more than half the air flow does not need to be decelerated for combustion and can go through the bypass instead. If the flow is split into the air needed for combustion and the air that can bypass, the amount of air that needs to be decelerated is also reduced. A smaller cross section is needed to reach the flame speed if only the air required for combustion is decelerated. This smaller diameter will provide a more compact and reasonably sized combustor and will have less loss in pressure because a large portion of the air does not need to be altered and can bypass the combustor as is.

This thought leads to the question of how best to split the air flow to the correct proportions with minimal pressure loss introduced. Several orientations were examined using a planar analysis. In reality, the geometries will be made of 2-inch-diameter piping. The analysis simulates the geometry as a per unit thickness in and out of the analysis plane. In order to relate the analysis to the true geometry a correlation factor is needed to relate the velocities calculated in Fluent to what they would be for the true round cross sections.

Mass flow is simply defined by a certain density of fluid passing through a given cross sectional area at an average velocity. For a planar analysis the flow is two dimensional meaning there is no cross section perpendicular to the flow direction. The flow is only analyzed in a single plane. Fluent calculates the mass flow in the same manner as above but does it as a mass flow per unit of cross section. This can be better understood by thinking that for a planar case the mass flow is calculated using just the width (D) of the cross section the fluid is flow through. The depth in and out of the plane is assumed to by unity (1).

$$A_{fluent} = D$$
$$m = \rho\, D\, V_{fluent} \tag{3}$$

The pipe that is desired to be represented is round and has a true non-zero cross section. In reality the pipe has a diameter (D) and a dimensionally accurate mass flow can be calculated.

$$A_{actual} = \frac{\pi D^2}{4}$$
$$m = \rho\, A_{actual}\, V_{actual} = \rho\, \frac{\pi D^2}{4}\, V_{actual} \tag{4}$$

The mass flow is what is being targeted in these analyses. Therefore the mass flow in the analysis can be set equal to the mass flow of the true geometry. By doing this a relationship is formed between the actual velocities with the velocities Fluent will calculate.

$$m_{actual} = m_{actual} = \rho\, \frac{\pi D^2}{4}\, V_{actual} = \rho\, D\, V_{fluent}$$

REDUCING TO

$$V_{actual} = V_{fluent}\, \frac{4}{\pi D} \tag{5}$$

Now it can be seen that the actual velocity can be calculated from the Fluent velocity simply as a function of the width D of the plane of flow. For a true round cross section the diameter is analogous and equal to the width of the plane in the Fluent geometry.

EQUAL ANGLE - SINGLE SPLIT

The air is extracted from the compressor in a 2-inch-diameter pipe. The simplest design would be to split the flow in two directions with a standard "y" shaped pipe design. For stabilized turbulent flow in a pipe, it could be assumed that if the flow is split into two equal diameter pipes at equal angles from the upstream pipe, the flow would be separated quite closely in half. For the operating point desired, 0.03 kg/s must be separated from the full 0.071 kg/s which is not half. The angles could be adjusted such that the correct proportions are naturally metered. This would be achieved by having the bypass pipe more in line with the upstream pipe than the combustion air pipe, seeing how more than half should be bypassed. However, there would be no margin or correction that could be used if it was later determined that the air is not being split properly. Or a different operating point may later be desired. Therefore, it was decided that the best option would be to have the angles split equally at 60 degrees from the centerline and then slightly throttle the combustion air side to adjust for the precise ratio desired. This throttling comes at the cost of pressure loss, so an analysis was run to determine how much throttle would be needed to proportion the flows correctly.

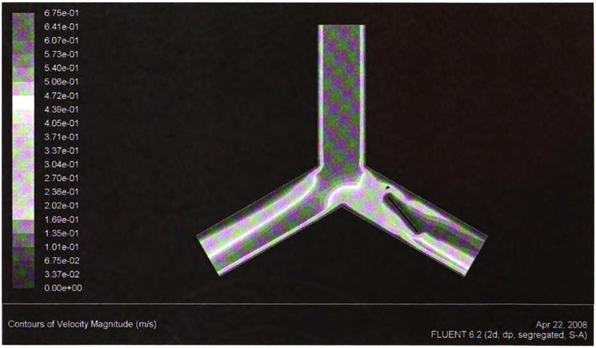

Figure 11. Equal angle – single split, velocity contour

The Fluent model was meshed using a planar type of analysis (Figure 11). The pipes were all 2-inches wide to be analogous with the true 2-inch-diameter pipes. A thin-walled obstruction was placed on the right side of one of the downstream pipes. This wall simulated the throttling effect of a butterfly valve that was intended to be used. It was found by Fluent simulation that the desired throttling of 0.030 kg/s through the combustion and .041 kg/s through the bypass was achieved when the butterfly valve was placed at roughly an angle of 9 degrees off from the full open position. Even though this was a planar analysis and the true geometry has round cross sections, it proved that a large pressure drop was not needed to achieve the desired flows. The analysis indicated that 68 percent of the total pressure was maintained as an average at the outlets relative to the inlet. Figure 12 shows a possible design that would utilize this type of flow separation.

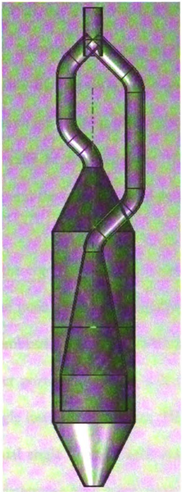

Figure 12. Equal angle – single split, hardware design

OFF ANGLE - SINGLE SPLIT

Due to the large cross sectional area needed in the combustion chamber to decelerate the flow it was thought for manufacturing purposes that the chamber should be axial and concentric to the inlet and outlet directions of the overall test rig. Specifically, this means that the chamber should be concentric to the flow before it splits into the combustion and bypass air. It should also be concentric to the flow after the two flows mix back together. By doing this, the overall size of the combustion chamber and bypass assembly can be minimized. The large chamber will be placed in the center and the smaller bypassing pipes can be easily routed around the outside of the chamber. Another reason for putting the chamber in the center was that the combustion process was expected to be at 1200 K. This temperature is close to the melting point of most steels used for this fabrication. By using the bypass air as a coolant the combustion chamber can be fired at temperatures above the melting point of steel because the bypass air can provide cooling to the outer walls of the chamber. Therefore, having the chamber inboard and the bypass flow around the outside would be preferred to cater to this design intent.

A configuration where the combustion air is maintained in a straight line and the bypass air is extracted from the main flow was considered. Less then half of the total mass flow needs to enter the combustion chamber. With the combustion pipe being axial with the upstream inlet a larger portion of the air will naturally enter the combustion chamber then will turn at an angle towards the bypass. This means that more throttling of the valve then the previous equal angle – single split design will be needed. A similar analysis as before was run to determine how much more throttling would be needed to achieve the 0.030 kg/s of mass flow through the combustion chamber (Figure 13).

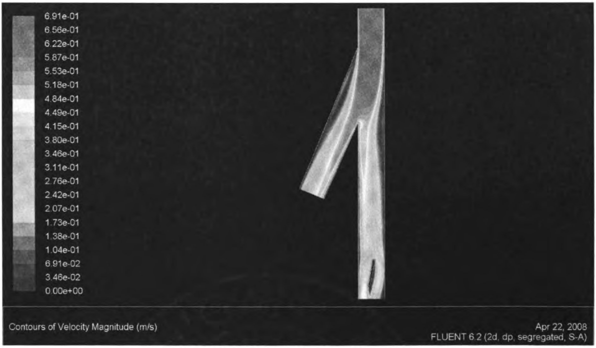

Figure 13. Off angle – single split, velocity contour

The analysis showed that the throttle valve must be placed at roughly 13.7 degrees off of fully open to achieve the desired throttling. This was 4.7 degrees more throttling than the equal angle – single split design. However, by analysis 81 percent of the total pressure was maintained across this geometry then the previous. It was worth noting that the axial placement of the combustion line required more throttling but yet less pressure was lost on average across the outlet ports. Figure 14 shows a preliminary design that would use this orientation.

Figure 14. Off angle – single split, hardware design

EQUAL ANGLE - DOUBLE SPLIT

For manufacture and air cooling reasons later discussed it was determined that the incoming air will need to be split three ways as opposed to two. The combustion air is still axial to the incoming air; however the bypass air is split at a 45 degree angle from the center line with in the same plane (Figure 15). Having the bypass air split away from the main inlet line at a steeper 45 degree angle causes less air to naturally bypass and more air to flow to the combustion. Again an analysis determined that the combustion pipe must be throttled to roughly 21.8 degrees to achieve the 0.030 kg/s flow rate through the combustor and 65 percent of the total pressure was maintained from inlet to outlets.

As expected the other .041 kg/s of air divides evenly through the two symmetry bypass pipes.

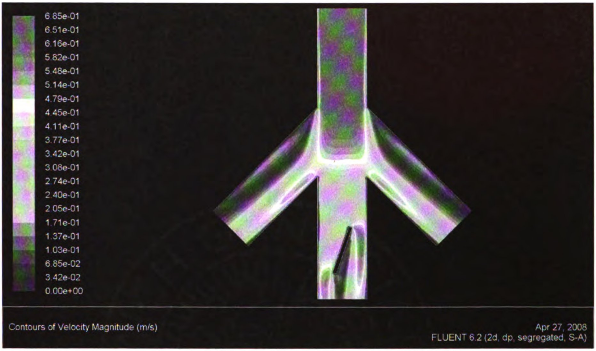

Figure 15. Equal angle – single split, velocity contour

DIFFUSION LENGTH

Now that incoming air has been split and metered into combustion and cooling bypass air, it was time to reexamine the combustion chamber design. This time the amount of flow in the combustor has been reduced from the total mass flow of 0.071 kg/s to only 0.030 kg/s. This means that the amount of mass flow that needs to be decelerated is less. The flame speed of propane has been calculated to be near 0.5 m/s, so any combustion chamber design should be able to create low speed regions capable of combustion [Kusner, 2006]. To assure this, a quick calculation can be done to set the cross sectional area of the combustor necessary to decelerate the flow to 0.5 m/s.

$$
\begin{aligned}
m &= \rho A V \\
m &= 0.030\ \mathrm{kg/s} \\
\rho &= 2.189\ \mathrm{kg/m^3} \\
V_{air} &= V_{flamespeed} = 0.5\ \mathrm{m/s} \\
A &= \frac{m}{\rho V_{flamespeed}} = 0.0274\ \mathrm{m^2} = 42.47\ \mathrm{in^2} \\
D &= \frac{4\sqrt{A}}{\pi} = 8.29\ \mathrm{in}
\end{aligned}
\tag{6}
$$

It was found that a 8.29-inch-diameter cross section correlates with the area needed to decelerate all of the air to be at or below 0.5 m/s. It would take an extremely long length of ducting at this larger area for all of the high velocity area to decelerate and fully develop at this slower speed.

In order for the entire combustor to be around 3 to 4 feet in length the diffuser section must be limited to roughly a foot in axial length. A first look was to use Fluent to determine the velocity profile of a simple conical diffuser assuming that the diffuser is restricted to 11 inches of axial length. It was determined that the geometry would be a 2-inch-diameter cross section diffused to an 8-inch-diameter cross section, which is the next nominal dimension larger than the idealized 8.29-inch-diameter needed to decelerate the air to at or below 0.5 m/s. This geometry was meshed and run in Fluent with the 0.03 kg/s mass flow expected in the combustor.

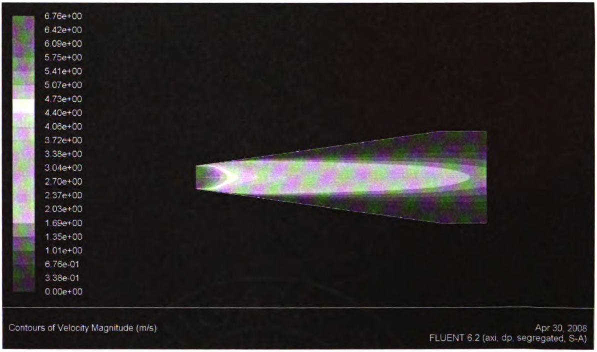

Figure 16. Conical diffuser, velocity contour

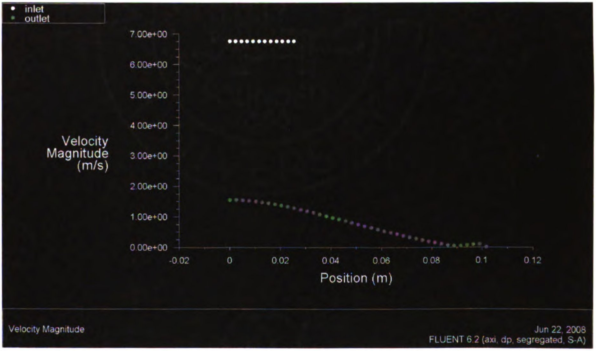

Figure 17. Conical diffuser, inlet and exit velocity profiles

The analysis was run as an axisymmetric model and was found to be turbulent flow. The inlet velocity through the 2-inch-diameter pipe was 6.76 m/s for the 0.03 kg/s mass flow. As can be seen in Figure 16 there was a significant region below 0.5 m/s. Figure 17 shows the velocity profiles across the inlet and outlet cross sections as a function of their radial distance outward from the central axis. The outlet shows that at a radial distance of 0.06 meters (2.4 inches) the flow of the air is at or below the 0.5 m/s target velocity. However as expected, the velocity down the center of the diffuser was found to be around 1.5 m/s which is 3 times the velocity desired. This analysis proved that slower velocities could be achieved with a shorter length of diffuser than the idealized 7 degree angle diffuser. The shorter length causes there to be a higher velocity jet down the central axis that does not decelerate as well. This axial length is acceptable, but more analysis must be done to determine what type of flame holder must be used to help the flame attach and stabilize in the regions with velocities above the flame speed.

LEADING CONE FLAME HOLDER

A seemingly logical place to locate a flame holder would be as far upstream as possible. This would allow for the best use of the available space. Placing the flame holder near the diffuser inlet should also help with diffusion. In the previous analysis there was a high velocity jet that did not fully diffuse. The flame holder will also serve as a splitter that will force the high velocity jet radially outward. Giving the incoming jet a radial component should cause the air to make better use of the expanding area and decelerate further.

The same diffuser geometry was analyzed with the addition of the flame holder described. The mass flow through the diffuser was again set to 0.03 kg/s (Figure18). The results showed a great reduction in the high velocity jet. As predicted, the incoming air quickly impinged on the conical shape of the flame holder and deflected radially outward. This resulted in a much larger region of velocity below the flame speed.

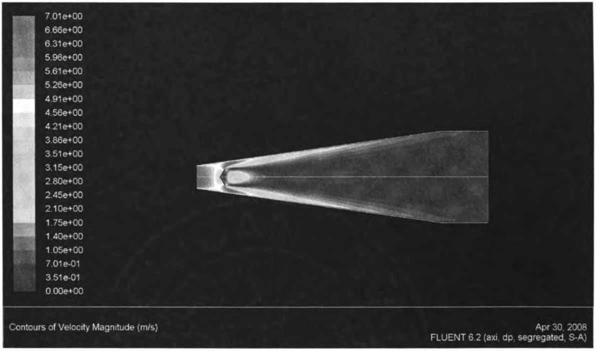

Figure 18. Leading cone flame holder, velocity contour

Figure 19 is a plot analogous to Figure 17 of the incoming and exit velocity profiles for the diffuser. The velocity profile across the entire outlet cross section is at or below the desired 0.5 m/s goal.

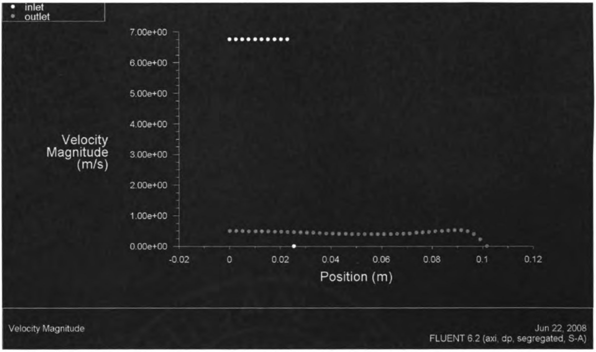

Figure 19. Leading cone flame holder, inlet and outlet velocity profiles

Having a leading cone type flame holder seemed promising. A closer looked proved otherwise. Showing the velocity magnitudes in vector form (Figure 20) indicates having the flame holder close to the inlet causes a large recirculation zone downstream. The air is rapidly forced outward and soon circulates back inward to fill the void caused by the flame holder. This inward swirling is noticeable throughout the entire length of the diffuser. In fact a large portion of the diffuser's inner flow has a velocity magnitude upstream. This upstream flow causes large amounts of shear against the rest of the downstream flow. This shearing created a large and unwanted pressure loss, only 11 percent of the total pressure was found to be maintained. The flame holder design proved to be capable of decelerating the flow but there needs to be more guidance with how the flow is directed and expanded.

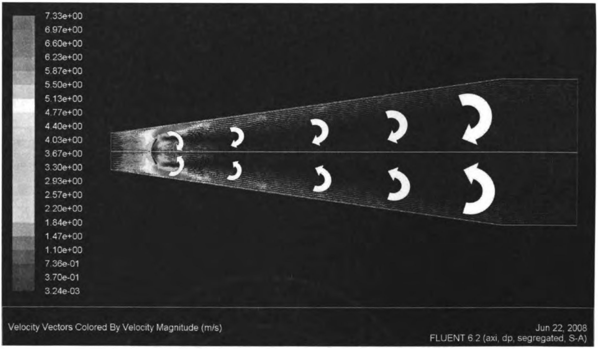

Figure 20. Lead cone flame holder showing recirculation zones

GUIDED DIFFUSION

At this point in the design process several attempts were made to make consistently shaped cones and round ducts. The large diameter round ducts were easier to make (Figure 21) with some minor but acceptable flaws. The tapered cones required a 2-inch-diameter on the smaller end of the cone. The tool being used to form the sheet metal was a simple hand cranked sheet metal roller. This roller could not provide the force needed to roll the sheet metal in such a tight 2-inch-diameter cross section nor, were the rollers small enough. Sending the parts out to be constructed professionally by a vendor with a power roller was not within the budget of this project. Therefore, an alternative option to using conical shapes was used.

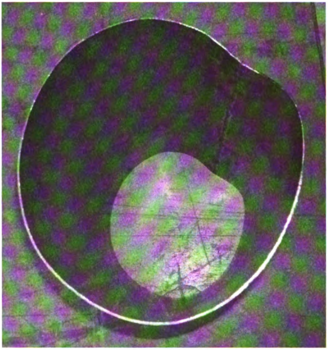

Figure 21. Straight duct section as fabricated

A solution was to use a square cross section. This way all of the sheet metal pieces would be flat pieces that could be cut with any saw capable and then seam welded together.

Now that it was known that square cross sections will be used, the analysis process changed slightly. The previous combustion chamber designs were axisymmetric and were analyzed as such in Fluent. The best way to represent square cross sections in Fluent without going to a full three dimensional analysis is with a planar analysis just like in the previous combustion bypass analysis. However this time, the planar mesh needs to be analogous to a square cross section. Again a correlation factor needs to be determined to relate the velocities calculated in Fluent with what the velocities would be in the true geometry. This factor was calculated in exactly the same manner as it was for a round cross section except the calculation for the true area is for a simple square and not a circular cross section.

$$A_{actual} = D^2$$
$$m_{actual} = \rho A_{actual} V_{actual} = \rho D^2 V_{actual}$$
$$A_{fluent} = D$$
$$m_{fluent} = \rho A_{fluent} V_{fluent} = \rho D V_{fluent}$$
$$m_{fluent} = m_{actual} = \rho D V_{fluent} = \rho D^2 V_{actual}$$
REDUCING TO
$$V_{actual} = \frac{V_{fluent}}{D} \tag{7}$$

As can be seen the true geometry velocity is analogous with the velocities Fluent calculates but on a per width basis. Fluent represents the velocity as the true velocity divided by the width of the plane the flow is passing through. It should be noted that the velocities and distances are true engineering units; in this case standard SI units are being used throughout all analysis.

As a bench mark, an analysis was run showing just the main diffuser along with the nozzle at the end of the combustor (Figure 22). This geometry was set up and run without any flame holder geometry to first establish how the flow reacts to the new planar analysis, which represents the new square cross sections.

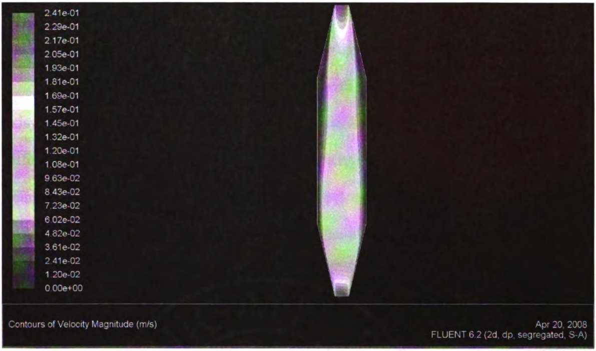

Figure 22. No guide vanes, velocity contour

The true geometry in mind is now an 8-inch by 8-inch square cross section as opposed to an 8-inch round cross section. In order to properly understand these plots the 8-inch width must be converted to SI units and used to divide out the Fluent velocities. Equation (7) shows what the Fluent equivalent velocity is for the 0.5 m/s flame speed goal.

$$\mathbf{D = 8\,in = 0.2032\ m}$$
$$\mathbf{V_{actual} = \frac{V_{fluent}}{D} = 0.5\ m/s}$$
$$\mathbf{V_{fluent} = V_{actual} = (0.5\,m/s)\,(0.2032\,m) = 0.102\,m/s} \qquad (8)$$

A velocity of 0.102 m/s shown in the plot above would be at or below 0.5 m/s in the true geometry. Just like in the round cross section analysis, there was a significant region below the desired flame speed temperature. Again there was a high velocity jet region along the central axis that does not decelerate as well as the flow further outboard.

As expected the square cross section design will need the addition of flame holders and/or guide vanes to encourage diffusion in a way that best conserves pressure.

Due to the recirculation zone issues with the abrupt flame holder in the axisymmetric analysis, it was thought best to look further into guide vanes to enhance diffusion. The guide vanes should gradually diverge the air entering the diffuser. By doing so, the flow should have a more even velocity distribution across the width of the diffuser, which will reduce shearing and the likelihood of recirculation zones.

Figure 23. Single guide vane, velocity contour

In the results above two simple guide vanes were placed at the inlet of the diffuser to represent a 4-sided, pyramidal shape in true geometry (Figure 23). This guide vane forced some of the flow outboard as expected, which resulted in much lower velocities in the main body of the combustor.

The idea to use guide vanes was taken one step further with another smaller divider added along the central axis (Figure 24). This divider served to diffuse and separate the center jet even further, resulting in much better diffusion than the single guide vane. In fact, after viewing the velocity profile (Figure 25) across the widest point of the combustor it was noticed that the velocity was below 0.5 m/s (analogous with 0.102 m/s as plotted in Fluent) for the entire cross section. This means that all of the air has been properly diffused, and it suggests that the propane flame would be capable of sustaining in this environment. The total pressure that was maintained across the combustor was quite similar for the single and double guide vane analysis, 34 and 33 percent respectively.

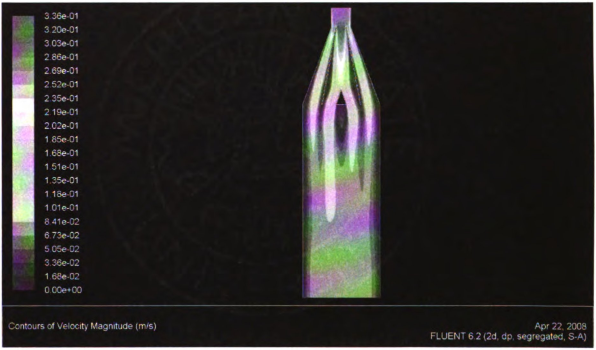

Figure 24. Double guide vane, velocity contour

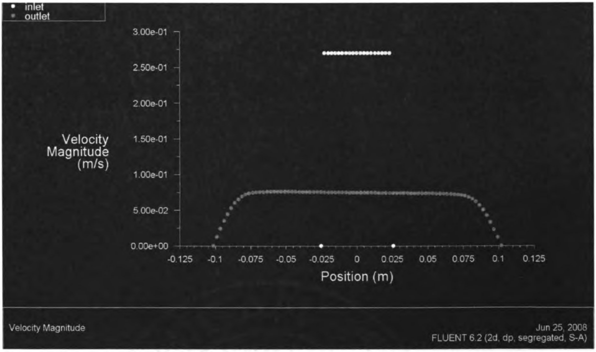

Figure 25. Double guide vane, inlet and outlet velocity profiles

CENTRALIZED FLAME HOLDER

Now that the velocity of the combustor was within appropriate limits it was time to bring some reality to exactly where the flame should be initiated within the chamber. Even though the previous guide vanes proved capable of decelerating the air they do not provide a location for the flame to initiate and attach. There needs to be a location for the fuel to be injected, then mixed with the air and finally ignited. A design that can achieve this would be a large pyramidal shaped flame holder along the central axis. Within this flame holder the propane will be injected. Small mixing could be placed through the holder to allow some air to mix with the fuel. Just before the fuel air mixture leaves the flame holder, it should be ignited. The tapered leading edge of the flame holder acts much like the guide vanes in the previous analysis to diffuse the incoming air outboard. The inside of the flame holder acts exactly as a fuel injector nozzle and also as the point of ignition.

The plot below shows the geometry of the flame holder. The analysis was again run with the expected 0.03 kg/s of air flow through the combustor. The leading contour of the flame created a large low speed zone well below the 0.5 m/s flame speed down the central axis of the combustor. In this region the flame will be ignited, and with these lower velocities it should be able to stabilize and attach down stream of the holder. Having the flame centrally located in the combustor allows for the air flow around the outside to act as a thermal layer and protect the outer wall of the combustor from the much hotter temperatures further inboard.

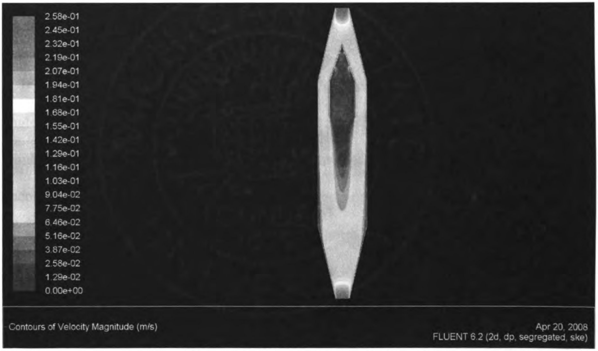

Figure 26. Centralized flame holder, velocity profile

This analysis solidified the final combustion chamber design (Figure 26). The final design will have a single flame holder along the center axis with a 4-inch cross section at its widest point. The top of the flame holder will taper upwards to the top where the fuel will be injected at the leading most point. The main body of the flame holder will be held in place by a 1 ½-inch pipe perpendicular to the flow direction.

Within this support pipe is where the pilot flame will be introduced to the inside of the flame holder. The fuel is injected within the flame holder and forced downward and must pass by the pilot flame to exit the holder. This will guarantee ignition. On the flame holder, 1/8-inch-diameter holes where drilled into the side walls of the leading guiding surface. These holes allow air to enter the flame holder over an evenly distributed area. This air is intended to premix with the fuel for better ignition.

As previously determined, the main body of the combustor will have an 8 inch square cross section. The diffuser leading in to the combustion region will also be a square cross section that increases from a 2-inch to 8-inch cross section just like the analysis validated.

The bypass air was determined to be split from the combustion air in the equal angle - double split analysis. This air will be introduced to the outer skin of the combustion chamber. Specifically, the bypass duct is a 9-inch square cross section that encompasses the 8-inch cross section of the combustion chamber. This creates a duct of 1-inch width around the perimeter of the combustion chamber wall. Directing the bypass air through the duct allows the air direct contact to the outer wall of the combustion chamber to provide a source of cooling. Cooling of the combustor is necessary to protect from the high flame temperature. Downstream of the flame holder and expected flame front, the bypass air enters the inner combustion cavity through large 1 to 2 inch holes in the wall between the bypass and combustor (Figure 29). At this point all the bypass air is reunited with the hot combustion gas. The introduction of the bypass will now further reduce the total air temperature to a desired 1200 Kelvin. At the bottom of the combustor is a nozzle similar to the analysis, which returns the flow back to a 2-inch diameter round cross section. These hot combustion gases are then ready to be expanded across the turbine. Figures 27 through 30 show the actual combustor as fabricated.

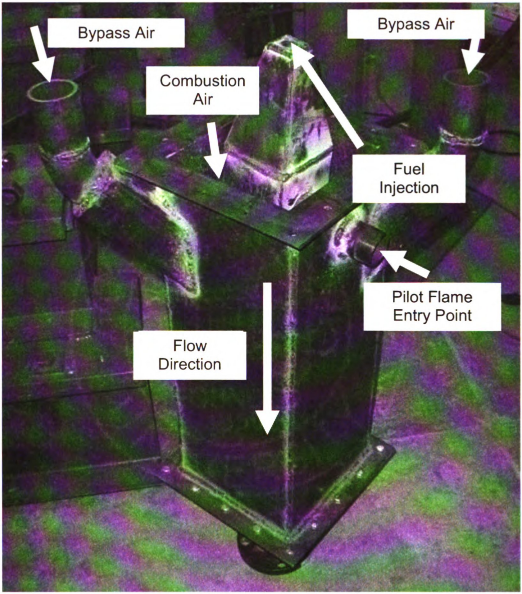

Figure 27. Centralized flame holder, front view

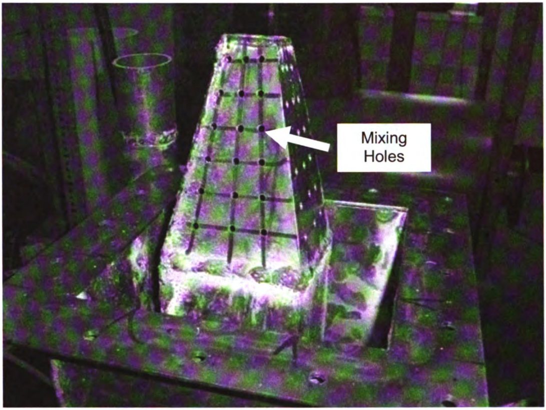

Figure 28. Centralized flame holder showing mixing holes

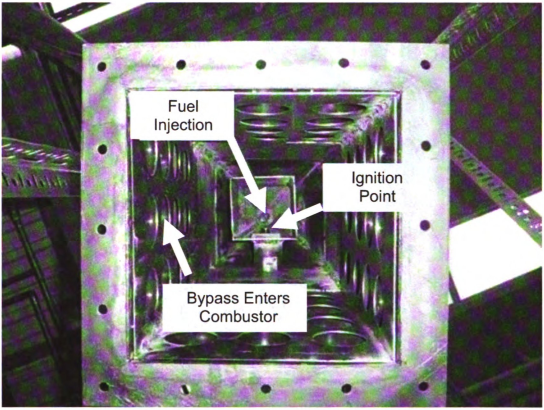

Figure 29. Centralized flame holder, bottom view

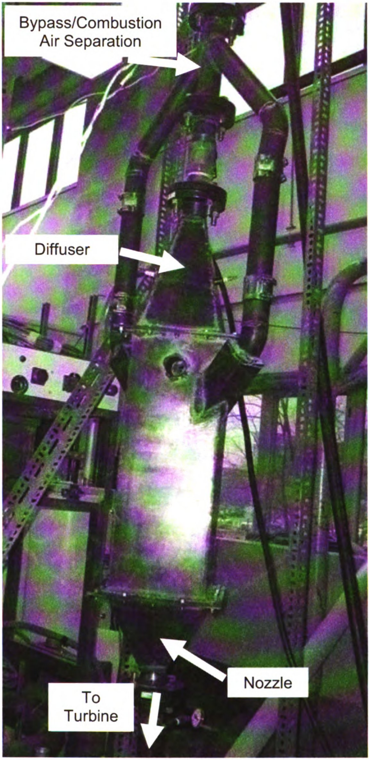

Figure 30. Centralized flame holder, full view

IGNITION SYSTEMS

INTERMITTENT IGNITION

The first combustor designs utilized commonly available automotive spark plugs. Initially these spark plugs were located on the outer wall of the combustor near the fuel injector (Figure 31). Like in an internal combustion engines, these spark plugs work when coupled with a distributor that builds up an electric charge and then releases when the circuit is closed. The discharge causes a hot electrical arc to span across the electrodes of the spark plug several times a second. It is desired that the flame in the combustion chamber be continuous and therefore the periodic ignition of the spark plugs is not ideal. The combustion chamber has been designed to have velocities at or below the flame speed of the propane fuel. This means that after the spark plugs ignite the propane there should be regions that can sustain the flame. At this point the flame should be self stabilizing and the spark plugs will only continue to ignite as a back up source of re-ignition in the case of the combustor flaming out.

This configuration shown in Figure 31, with the spark plugs being located on the outer walls of the combustor was tested. This orientation proved inadequate to ignite the propane. The velocity of the incoming air quickly carried the fuel downstream before it had a chance to diffuse outwards and be ignited by the plugs.

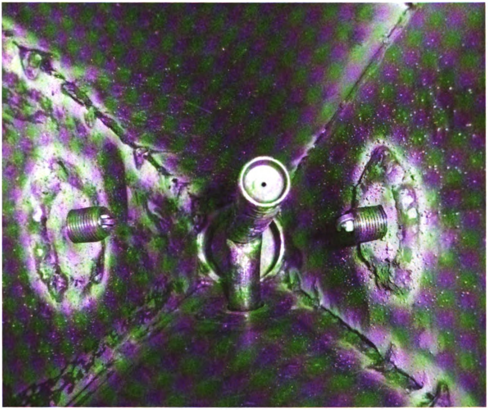

Figure 31. Spark ignition with spark plugs on outer wall

The solution to this was to place a single spark plug directly downstream of the fuel injection after the fuel has a chance to mix with some air (Figure 32). Even with the spark plug running, it took several seconds for the propane to ignite and sometimes it never did. The times it did ignite there was a loud bang followed by a large release of pressure. This quick release robbed the system of air leaving no chance of continued combustion afterwards. For the spark plug to properly ignite the fuel, several things need to happen. Not only does the fuel need to mix properly with incoming air but the ignition must start from the small electrodes of the spark plug. After these large explosions were witnessed it was apparent that the spark plug was not a large enough source of ignition. The ignition source should provide combustion instantaneously after fuel delivery. It is dangerous to have unburned fuel accumulate in the chamber.

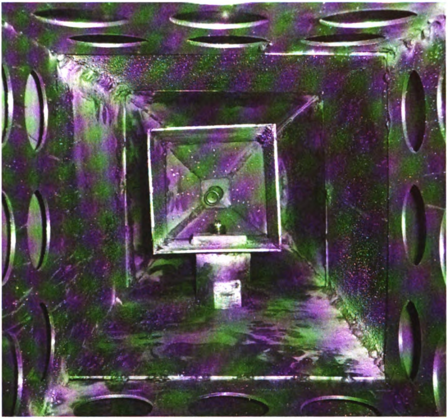

Figure 32. Spark ignition with spark plug inside flame holder

CONTINUOUS IGNITION

It was decided to reject spark plugs as an ignition source all together. A pilot flame like those found on residential furnaces was considered. The main issue to consider with using a small pilot flame is that it is susceptible to being blown out by the high velocity air. This means that the flame itself must be robust and able to handle the velocities calculated in the chamber design analysis. Many types of pilot flames were tested but the best source turned out to be MAPP gas mixed with pure oxygen. A small bottle of MAPP gas and oxygen was able to be obtained from a local hardware store as a package set (Figure 33). This set was marketed with a small metal torch that regulates the mixture of MAPP gas and oxygen. When a small amount of MAPP gas was mixed with a large amount of oxygen, a very small blue flame was achieved. Premixing the

MAPP gas with oxygen makes this pilot flame robust. The MAPP and Oxygen mixture was already properly set by regulating the two bottles. This allows the mixture to ignite as soon as it is released from the nozzle tip and attach itself to the end of the nozzle. The pilot flame attaching to its nozzle reduces the likelihood of blowing out.

The flame torch was located downstream of the fuel injector in the same location as the single spark plug was first attempted. The pilot flame was tested in the combustor and proved to have no issues with igniting the propane. The fuel was instantly ignited when introduced into the chamber and remained ignited. Now that stable combustion was reached, the mass flow was varied across a broad range to test for flame stability. The pilot flame was even removed and stable combustion was still maintained. This proves that the air in the combustion chamber was indeed diffused and decelerated enough to allow for a stable flame to exist without blowing out. It was decided that this pilot flame will be the ignition source for future testing.

Figure 33. Ignited pilot flame with MAPP gas and Oxygen gas kit

STARTING SYSTEMS

GATE VALVE

Combustion can not start the engine from rest by itself. The engine was started with a high pressure air line. This air line was used to force air through the compressor and then expand across the turbine. Under normal operation, a compressor pulls forward on the shaft. Forcing the air through the compressor in such a direct manner would cause large stresses on it by pushing it aft instead. Severe loading on the compressor-turbine couple was believed to be the result of previous testing, which caused the shaft to break at high velocity (Figure 34). Breaking the shaft caused the compressor to surge forward into the inlet shroud, which ruined the blades.

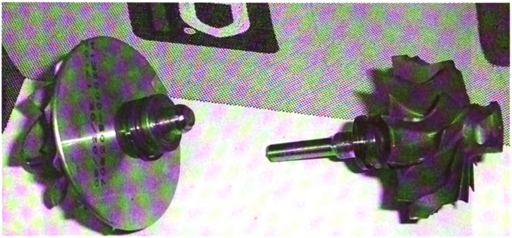

Figure 34. Broken compressor-turbine shaft

At this point an identical compressor-turbine-bearing assembly was purchased. To prevent this problem from happening again, a one-directional swinging check valve was designed (Figure 35). This valve was placed in the engine flow between the compressor and turbine. The swinging gate only allows for the starting air to flow downwards towards the turbine. Now the turbine drives the compressor which in turn pulls in ambient air. As the compressor is spooled up to speed, the starting air will be reduced slowly and the swinging gate valve will progressively close. This will allow the compressor fed air to cycle through to the turbine, thus the engine begins to run under its own power.

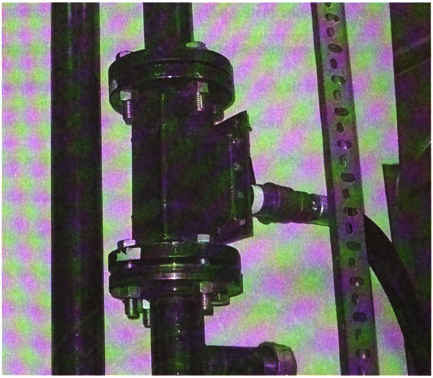

Figure 35. Swinging check valve used for engine starting

During testing it was found that the gate valve actually allowed the high pressure starting air to flow in all directions. This means that a significant proportion of the injection air was back-flowing and pushing against the compressor. Enough of the flow went towards the turbine to spin the turbine-compressor couple in the correct direction. However, the system was never able to accelerate up to the operating point because the compressor was not capable of developing the pressure to push back against the 90 psi start up air pressure. The startup air was capable of back-flowing due to the design of the air injector itself. The air was injected perpendicular to the intended flow direction. The idea was that this perpendicular flow would actuate the swinging gate to open when the start up air was being injected and direct the flow downward towards the turbine. By testing it was found that the injected air was quickly stagnated because of the recirculation zones caused by being injected perpendicular to the intended flow. This allowed the static pressure to build up in the system and the air was able to back flow around the cracks in the swinging gate towards the compressor and out the inlet.

The combustor was located directly below this air injector. The large increase in area was also an issue with the back-flowing. From the injector and upstream to the compressor is a consistent 2-inch-diameter cross section. Downstream of the injector towards the turbine the combustor drastically increases to an 8-inch square cross section. Therefore, the start up air that properly flows towards the turbine must first be expanded to this larger cross section and then contracted back to the 2-inch-diameter pipe before entering the turbine. At the location of the air injection it is easier for the air to flow out towards the compressor than it is to exit out the turbine properly. It was concluded that not only does the geometry of the start air injector need to be improved but also its physical location in the engine.

TURBINE NOZZLE

In attempting to correct these issues a new air injector was designed. Due to the expansion issues caused by the injection air flowing through the combustor, it was determined that any new design should be located downstream of the combustor just before the turbine inlet. As a design constraint any starting system must be able to handle the high temperatures caused by the combustion. Another issue was the high static pressure build up that caused the air to back flow in the check valve design. Two modifications have been made to remedy this. First the air will be injected in the downstream direction towards the turbine inlet, concentric and parallel to the intended flow of the engine. This prevents any recirculation zones and forces the air directly towards the turbine with minimal static pressure build up that would encourage back flow. Also the air was injected through a 3/8-inch diameter pipe. The old design injected the air through a 1 inch diameter pipe. This smaller diameter forces the air to enter the system at a higher velocity which increases the dynamic pressure, and as a result drops the static pressure that previously led to back-flow.

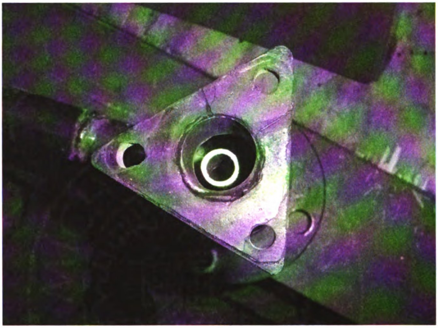

Figure 36. Turbine nozzle used for engine starting

This turbine injector nozzle showed several improvements over the previous design when tested (Figure 36). The turbine quickly turned over with less air flow than the previous design took for the system to start spinning. The compressor instantly started pulling in air from ambient. The previous injector design caused air to actually flow against the compressor. Therefore, this design clearly showed that it was capable of accelerating the air in the proper flow direction.

There were other issues when using this design to start the engine. For the first test runs the compressor-turbine couple was sped up to 70,000 rpm with the starting air. At this point the combustion process was initiated. It was expected that the introduction of the combustion should correlate with an increase in rpm, yet the rpm remained constant even in the presence of the combustion. This suggested that the added energy provided by the combustion did not correlate with an increase in turbine speed. A hypothesis is that the turbine inlet nozzle may be choked.

The air was injected into the system through the high pressure line just upstream of the turbine inlet nozzle and was quickly expanded across the turbine. The energy that was extracted spun up the compressor. The compressor started pulling air from ambient and passed it through the combustor. When mixed with fuel and burned, the air was fed downwards towards the turbine. When this compressor fed air reached the turbine, it combined with the starting air which was also being fed just upstream of the turbine. This was potentially why the system was not able to increase in speed with the added combustion.

The turbine nozzle was believed to be choked because at the turbine inlet the mass flow of the starting air and also the air from the compressor must pass. This would explain why the added energy from the combustion process could not accelerate the turbine; the turbine inlet was already passing as much air as it possibly could.

Having two sources of mass flow when starting also created the issue of transitioning from the starting air to the compressor fed air when stable combustion was reached. When the starting air is removed, the operating point of the engine would be quickly reduced causing a loss of stable combustion.

The proposed solution to these issues was another engine starting system redesign. The new design should only force one source of air mass flow through the entire system. This eliminates the issue of choking the turbine inlet nozzle because the amount of air used for starting the engine is also the air being used for the combustion.

This means that at any given time the mass flow of air across the compressor would be the same as the mass flow of air across the turbine, conserving continuity. The new design should also aim to improve the process of reducing the starting air from the system when the combustion process is sufficient to run the entire system.

COMPRESSOR NOZZLE

A better design then the turbine nozzle starter was to inject the starting air through the inlet of the compressor but in a safer manner than first attempted. This solves the issue of having two mass flow sources feeding into the system because whatever would be forced through the compressor would be pushed through the turbine. This also allows for the total flow of air through all points of the system to be measured, and reducing the need for two flow measuring devices to one. Also the transition period from starting air to solely compressor fed air can be addressed by the axial location of the starting air injection nozzle.

The air injection nozzle will be lined axially with the compressor inlet, but will be placed several inlet diameters upstream of the compressor inlet. This will allow the compressor to start pulling in air under its own power as the starting air is slowly reduced, thus easing the starting air removal period.

Two unknown variables were how far upstream the starting air injector should be placed from the inlet of the compressor and at what diameter the starting air should be released from the injector nozzle. The distance of the injector from the inlet of the compressor will be a trade off between the amount of starting air that makes it into the compressor and the amount of air that misses the inlet completely. It could be guaranteed that all of the starting air enters the compressor by placing the injector nozzle close to the compressor. The trade off would be that the compressor will not be able to pull in air from ambient under its own ability and the injector would create a blockage to the compressor when the system is running under its own power. The other design constraint was the diameter of the jet from the starting air injector. The diameter of this jet will be driven by the exit diameter of the air injector. If the diameter of the injector exit is large, the exiting flow will be slower and as a result have a much higher static pressure. If the nozzle is designed with a smaller diameter, the flow will exit the nozzle faster and have a larger dynamic pressure component. It was desired that the air enters the compressor in a way that is similar in state to the air that would be entering the compressor during stable combustion when at the operating point. An analysis of the starting air injector nozzle was conducted to answer these unknowns.

The first decision was to determine at what diameter to eject the air from the start air nozzle. The inlet of the compressor is 54-mm in diameter, which is roughly 2 inches. When air is released from the high pressure line it will form a high velocity jet. The cross section of this jet will increase in diameter as the jet progresses axially, implying that the jet should be released at a diameter smaller than the inlet diameter of the compressor. To ease manufacturing a 1-inch-diameter schedule 40 pipe was chosen as the best choice among readily available pipe sizes.

The last step was to determine at what axial distance from the compressor inlet the 1-inch diameter pipe should be placed. To achieve this, an axisymmetric analysis was modeled to represent this geometry (Figure 37, 38). On the left a nominal mass flow of 0.05, 0.075 and 0.1 kg/s were ejected from the high pressure nozzle and allowed to freely expand at atmospheric condition. On the right side of the mesh the geometry of the compressor inlet was modeled. The outer perimeter of the mesh was modeled with outlet boundary condition, creating a situation where the air ejected from the high pressure nozzle was free to entire the compressor inlet or exit the analysis zone all together. For the different mass flow inputs from the high pressure line the amount of air that entered the compressor inlet and the amount that missed were recorded.

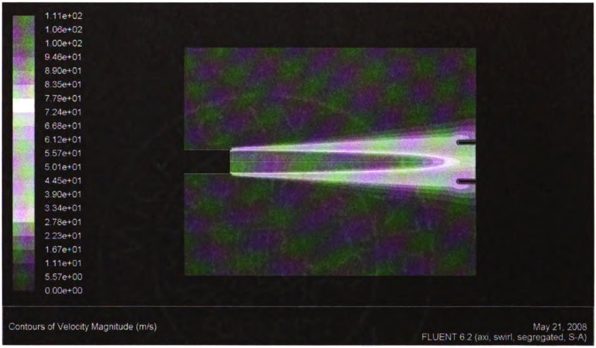

Figure 37. Compressor start nozzle, velocity contour

The axial position of the high pressure line from the compressor inlet was varied from 8 to 14 inches in 2-inch increments. It was found that the percentage of mass flow that leaves the high pressure nozzle relative to the amount that reaches the compressor inlet does not vary greatly based on the amount of mass flow (Figure 39). In contrast it was found that the axial position of the high pressure line has a dramatic effect. At 11.5 inches the amount of mass flow that leaves the high pressure nozzle was equal to the amount of air that reaches the compressor inlet. At an 8-inch axial position 125 % of the mass flow provided by the high pressure line reaches the compressor inlet. This was possible because the high velocity jet was entraining the stagnant air from ambient and accelerating it towards the compressor. This added mass flow comes at a penalty. The high velocity jet was decelerated when it sheared with the stagnant ambient air, which in turn was being accelerated. However, this was an acceptable penalty. The high pressure line can provide more than enough air and it is more important to have the nozzle as far away from the compressor as possible. Therefore, it was determined that setting the 1 inch diameter nozzle 11.5 inches from the compressor inlet in a concentric manner is the best balance.

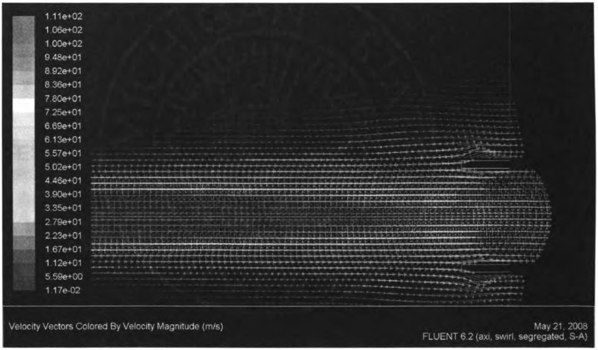

Figure 38. Compressor start nozzle, velocity vectors

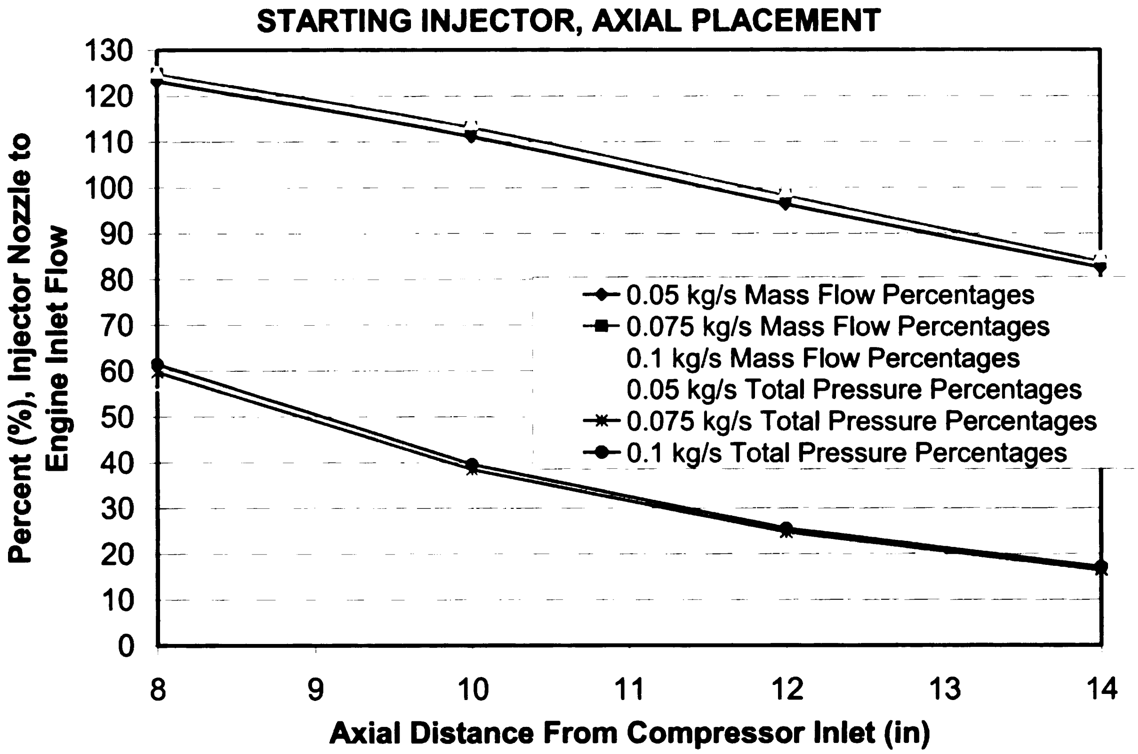

Figure 39. Compressor start nozzle, operating characteristics

TESTING CONFIGURATIONS AND RESULTS

DATA ACQUISITION SYSTEM

The objective of this thesis was to design a test rig capable of running meaningful and repeatable test runs. There are three major configurations of this test rig; the first was to run the combustion system solely by itself, then run the combustor with the turbocharger, and then run the combustor with the wave rotor.

A final fourth goal was to run the combustion chamber coupled with the turbocharger and the wave rotor to test and validate these technologies. A logical step was to first design the combustion chamber and validate that it was capable of the operating ranges desired to test the turbocharger and wave rotor.

The mass flow rate desired for both the turbocharger and wave rotor was nominally 0.071 kg/s. The combustor along with any of the acquiring devices should be capable of a broad range above and below this nominal operating point. Several mass flow measuring technology types were considered. Devices that are capable of measuring mass flow directly are quite expensive. This is because they rely on multiple types of technology within the same device to measure and then calculate the mass flow rate, usually by measuring temperatures and inertial effects. A direct mass flow sensor was deemed too expensive to remain within the scope of this project. As a substitute a volumetric flow sensor was purchased with intent of externally calculating mass flow. Volumetric flow rate is density dependent and therefore not a consistent way to measure compressible flow. Mass flow is simply the volumetric rate of the flow multiplied by its density. To obtain the density of the flow ideal gas was assumed. This was a strong assumption because the medium being dealt with was air which should remain well under supersonic speeds, certainly at the points where flow rates will be measured. The ideal gas law states that density is simply a function of temperature and static pressure. This means that the static pressure and the temperature of the flow, coupled with the volumetric flow rate, will yield the mass flow of the air.

$$\mathbf{V}\rho = \frac{\mathbf{V}\,P_s}{R\,T} = m \tag{9}$$

Even though three devices will be needed with this method, a volumetric flow sensor, a pressure transducer and a thermocouple, the total cost to achieve the mass flow rate was still less expensive then a direct mass flow device. As can be seen in Figure 40 the temperature and pressure devices must be sufficiently close to the volumetric flow meter because the volumetric flow changes throughout the system.

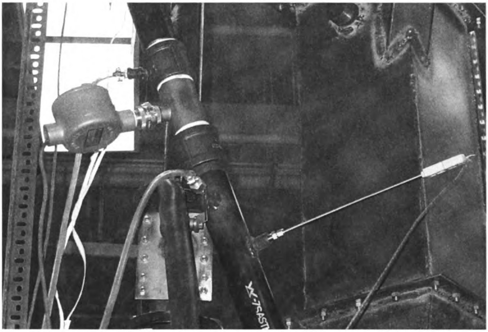

Figure 40. Data acquisition devices: volumetric flow, pressure and temperature

A simple electric pressure transducer capable of outputting a signal proportional to the pressure reading was used to measure pressure. Special care was taken to make sure that only the static component of the flow's pressure was measured. This was achieved by drilling a small hole perpendicular to the flow through the wall of the piping. It was important to ensure there are no burrs sticking inwards into the flow. These burrs could cause some of the dynamic pressure of the flow to build up with the static pressure tap and give a false reading.

The temperatures were measured by ungrounded K-type thermocouples. The ungrounded feature of the thermocouple was desired due to the large amount of other sensors that may electrically interfere with the readings. It also eliminated the need to isolate the thermocouples from the metal test rig. Not only was the temperature needed at the point of mass flow calculation but the maximum temperature of the combustion was a crucial factor as well. It must be monitored not only for matching the thermodynamic cycle desired but to monitor the safety of the rig itself. The maximum combustion temperature was expected to be 1200 Kelvin which is relatively close to melting limits of the materials used to manufacture the rig. The K type thermocouple was chosen because it has a range very close to the 1200 Kelvin expected in combustion but is also sensitive enough to measure the lower temperatures expected at the mass flow calculation points (Figure 41).

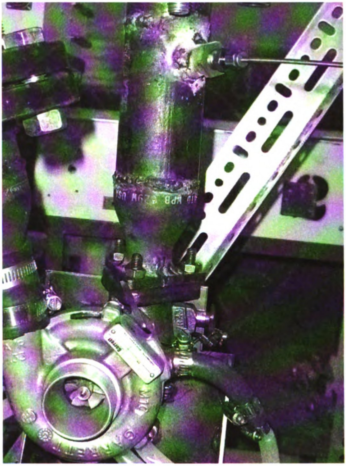

Figure 41. Turbine inlet temperature thermocouple

Close control and monitoring of all testing was required. To meet this all of the data must be captured and viewed in real time during testing. Therefore all of these devices are managed by the use of a data acquisition board and fed to a computer for the operator's viewing (Figure 42). The computer managed these data inputs by a data acquisition software program that was specifically tailored for this testing.

The data acquisition board has embedded cold junctions so the thermocouples can be wired in differential mode with no external reference temperature needed. This board was also capable of handling the flow sensors as well as the pressure transducers.

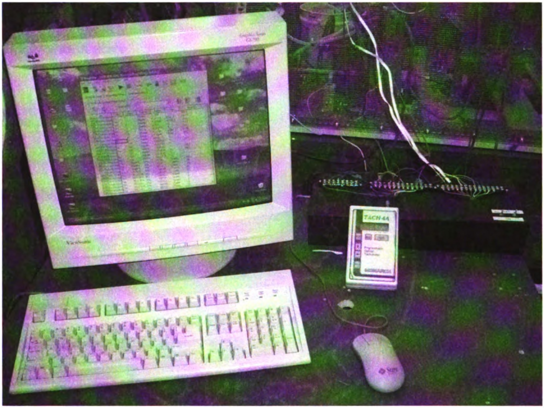

Figure 42. Data acquisition board and control computer

COMBUSTOR TESTING

Previous testing of the original combustion chamber proved to be intermittent and therefore prohibitive of further testing. This lead to the logical step to test this new combustion chamber design uncoupled from the turbocharger or wave rotor. This way the air delivery to the combustor was not dependent on the turbine-compressor assembly for the turbocharger case or on the shock wave propagation for the wave rotor. Doing this, any issues that were present with the combustion process could be isolated from the rest of the system and better examined. With no compressor the air must be continually fed to the combustor. This air will be supplied by the high pressure line available in the test cell. The volumetric flow meter along with the thermocouple and pressure transducer will provide the mass flow rate. Another thermocouple downstream of the combustor will output what would be the inlet turbine temperature if the turbine-compressor assembly was in place.

The goal of this testing was to prove that the combustor was capable of sustaining a flame at the desired operating point of 0.071 kg/s for the turbocharger and waver rotor devices. The temperature of the flow must be elevated to the turbine inlet temperature of 1200 Kelvin. If this flow rate and temperature can be achieved, the test would be a success.

The results of the first test can be seen in Figure 43. The first step was to light the MAPP gas pilot flame. Once this was done, the air flow was slowly brought up to the final mass flow operating point. At this time the fuel was added. Instantly, a low roar was distinguishable followed by a sudden rise in the inlet turbine temperature thermocouple. After this ignition was detected, the fuel source was throttled until the desired 1200 Kelvin was achieved. This operating point was held for roughly a minute and then the fuel was removed from the system. The temperature quickly dropped and the air flow was returned back to zero. The test was a success. After looking at the data it was apparent that the chamber was not only capable of sustaining a flame, but a reasonable tolerance was able to be held on the inlet turbine temperature at the final operating point.

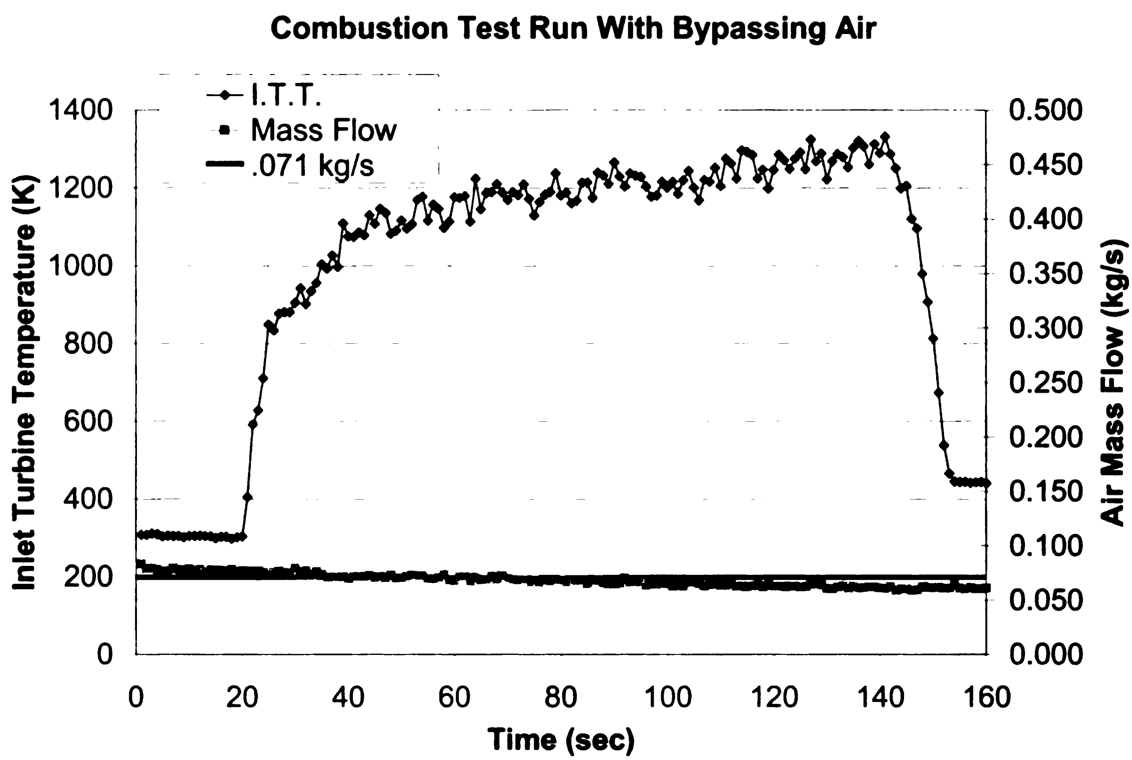

Figure 43. Combustor only test with bypassing air present

After executing the first test run it was noticed that the chamber seemed to have no trouble holding a flame without propagation. In the first test 0.040 of the 0.071 kg/s mass flow was bypassed around the outer shell of the combustor. Due to the positive results of the previous test a logical second test would be to see if the bypass air was truly necessary. The bypass piping was removed, which forced all air to enter the combustor. This increased the likelihood of flame propagation and flame-outs; yet there is a large benefit to this test. If the combustor is found to be capable of handling all of the flow, then the bypass system can be removed permanently. As previously analyzed, the bypass system comes at the price of having to throttle the flow. This loss in pressure could be eliminated if the bypass is not needed. The bypass air does provide an amount of cooling to the outer skin of the combustion chamber. When running without the bypass, the temperatures must be closely monitored to protect the test rig.

This test was run in the same manner as the previous, lighting the pilot flame first. The air was slowly brought up to the operating flow rate and then the ignition was started. Like the first test the ignition instantly responded showing a quick rise in temperature. Again the fuel was throttled back until 1200 Kelvin was achieved. This temperature and flow was held for roughly 2 minutes. For shut down, the fuel was first removed. When a temperature drop was noticed, the air flow was slowly reduced to zero. The test results (Figure 44) showed another success. The temperature proved to be stabilized around 1200 Kelvin with no large fluctuations. This test suggests that the ignition system and the geometry of the combustion chamber were capable of handling the entire mass flow rate without bypassing any air. The outer skin of the combustion chamber did not appear to be damaged by any elevated temperatures nor did any thermocouples indicate violent temperatures due to the lack of bypass air. At this point it was determined that the bypass will not be needed. This means that these pressures can be maintained. The combustor chamber analysis was rerun (Figure 45) with the full 0.071 kg/s mass flow to document these higher velocities experienced with no air being bypassed. As expected, increasing the flow rate through the combustor corresponded to a proportional increase in velocity.

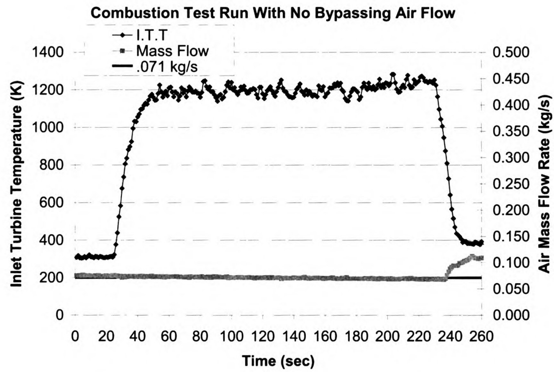

Figure 44. Combustor only test without bypassing air present

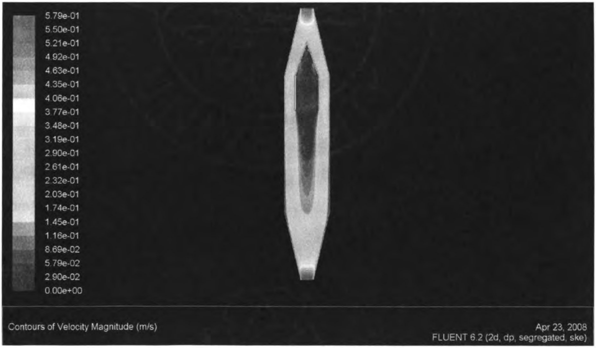

Figure 45. Centralized flame holder operating at .071 kg/s, velocity contour

GAS TURBINE ENGINE TESTING

Once the combustion chamber had proven sufficient for the operating point and heat input the next step was to test with the turbocharger intact. As discussed in the starting-systems section, all of the systems tested were capable of delivering air to the combustion chamber. However, none of the starting system designs were able to take the engine up to full operating point and run sustainably without starting air assistance. When attempting to start the engine, there was always a point when the compressor-turbine couple would not accelerate any further regardless of the amount of energy that was added by increasing the fuel flow. At this limiting point the compressor would rapidly make a popping noise. This noise would be followed by a slight drop in flow rate through the system. The suspicion was that the surge margin of the compressor was being reached and the noises being heard was the phenomenon known as compressor surge.

Surging is the limit of a compressor to move and compress air. As air flows through a compressor, its pressure rises. This means that the air downstream of the compressor has a higher total pressure than the upstream air. There is a limit to how much flow a compressor can handle aerodynamically. As more air passes through the compressor, the aerodynamic losses are increased due to the higher velocities. Eventually a point is reached when these losses are so great that the compressor cannot compress any more flow. When this occurs, the high velocity air will stagnate downstream of the compressor. This stagnation causes the static pressure to rise downstream of the fast spinning compressor blades. This air can now escape out the front of the compressor because it is not being pushed downstream by newer compressed air. When this air escapes forward, it slams the compressor rotor forward violently against the volute.

To explore if this surging was taking place, the engine could be run in an open-loop configuration. By open loop it is meant that the compressor and turbine will not see the same air flow. The combustor will be routed with the turbine side of the turbocharger. Air from the high pressure line will be fed through the combustor then mixed with fuel before being fired. Finally, the hot gas will be expanded across the turbine and leave the system as exhaust. This process will in turn spin up the compressor. Normally the compressor would be feeding air into the combustor. For an open-loop test this air will be dumped to the atmosphere. For the engine to be able to run in a closed-loop system the mass flow of the compressor must match the mass flow of the turbine. This open-loop test will be able to measure the amount of flow through the combustor-turbine side which is being driven by the high pressure line manually. The test will take the combustor-turbine side up to 0.071 kg/s and 1200 Kelvin then witness if the compressor is capable of producing the 0.071 kg/s mass flow rate needed to run the system in closed loop.

The first open-loop test was to see how much mass flow could be generated by the compressor when the turbine was being driven by the 0.071 kg/s flow rate. This test was without combustion. Due to losses in the system it was not possible for the compressor to be able to move as much air as the turbine without any additional energy input from the combustion. This test was intended to serve as a bench mark for the true combustion tests.

With no combustion, the process for running this test was quite simple. The mass flow through the combustor-turbine couple was slowly increased. During testing the mass flows of both the compressor and turbine side are being acquired. As seen below (Figure 46) a plot was generated with the resulting data that shows the difference in mass flow between the turbine and the compressor sides. With no combustion the compressor seemed only to be able to output roughly half the mass flow that was passing through the turbine side. This mass flow mismatch is the energy loss the combustion must be able to overcome. It should also be noted that the popping sound believed to be surging was not present during this test. This could be because the compressor was operating at a much lower flow rate than it was when combustion was present during the attempts to start the engine as a closed-loop system.

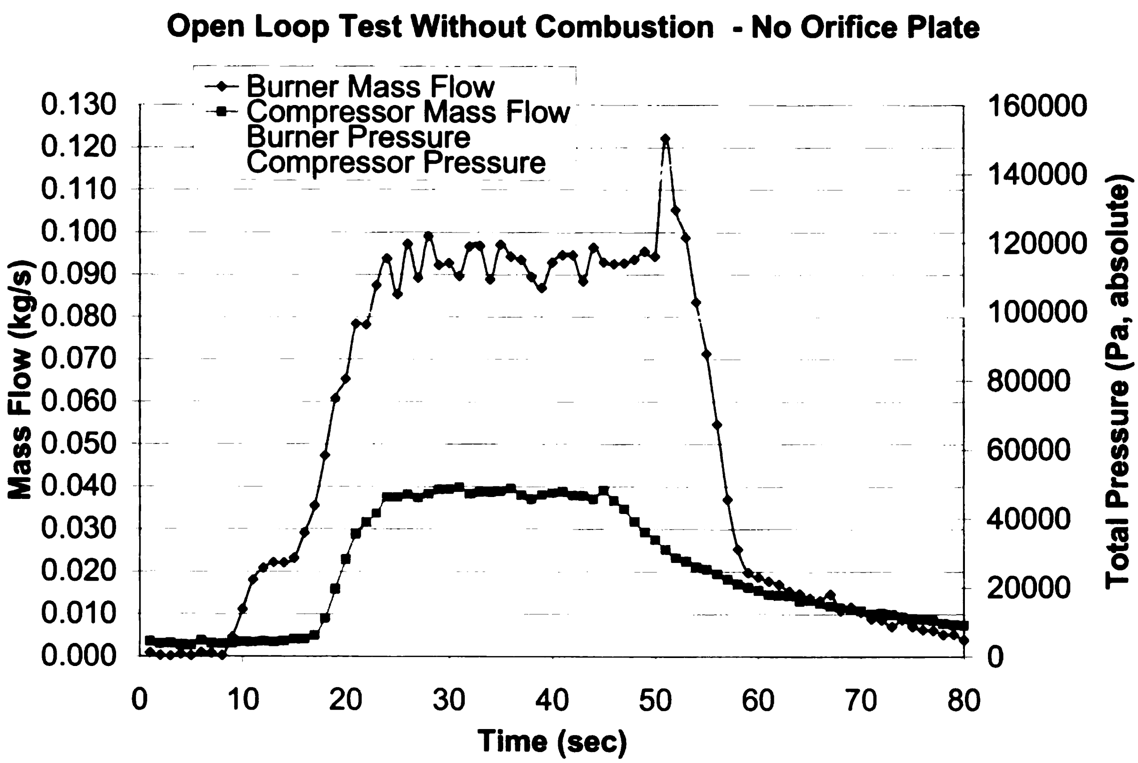

Figure 46. Open-loop turbocharger test without combustion, mass flow and pressure

As can be seen by the results of the first test, there was no significant rise in total pressure from the compressor. The total pressure in the turbine side was found to be roughly 30 % higher than the pressure in the compressor side. In fact the compressor did not appear to raise the pressure of the incoming air much above atmospheric pressure. This low pressure would be an issue when attempting to run the rig in a closed system configuration. With no combustion it takes a greater amount of flow and pressure to yield the same results then if more energy was present in the form of heat addition.

Due to this lack of pressure it was determined that an orifice plate should be installed at the compressor discharge. By forcing the compressor to discharge through a smaller orifice the compressor must work harder and develop a higher static pressure. This should match the pressure through the turbine side better.

The normal discharge area for the compressor was a standard 2-inch-diameter pipe. Several iterations and small tests were run to determine that a 1-inch-diameter orifice at the compressor discharge would be a conservative starting point. Using an orifice plate that was too small could cause damage to the system. If too much flow was blocked, the compressor would not be able to discharge air as quickly as needed. This would result in a spike in pressure which could be catastrophic to the rig. If the compressor does not respond as expected with an increase in static pressure, another smaller orifice plate could easily be installed instead.

The next test with an orifice plate in place was run exactly like the previous (Figure 47). The air was increased until the desired flow rate was achieved. Meanwhile static pressure, temperature and mass flow were being acquired for both the compressor and turbine. As can be seen in the results the compressor still fell short of matching the mass flow through the turbine. This was expected like it was in the previous test due to the absence of combustion. What was most notable was that the compressor side seemed to still have little pressure rise even with the introduction of the orifice plate. This suggests that the orifice plate should be smaller than the 1-inch-diameter. However the introduction of combustion could change this. It was decided that testing with combustion should be run before a decision to reduce the orifice plate would be made.

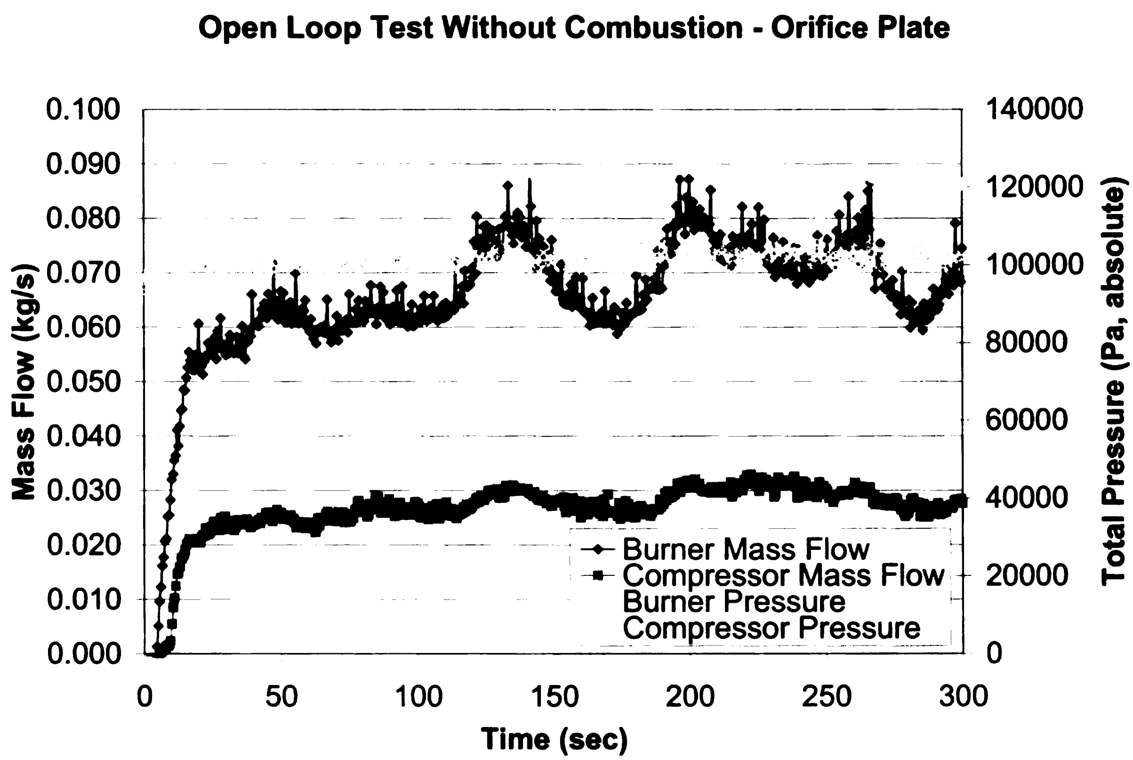

Figure 47. Open-loop turbocharger test without combustion, mass flow and pressure

The next test was to add in the combustion process and determine if the compressor was capable of producing a matching flow rate to the combustor-turbine side. The introduction of combustion compensated for the mechanical losses in the system and allowed the compressor to meet or exceed the mass flow rate of the turbine. In order for the system to be run in a closed loop as an engine the pressure created by the compressor should also meet or exceed the turbine side to maintain flow in the correct direction.

The testing process was quite different when combustion was taking place. First, the pilot light was lit. Then the air being forced through the combustor-turbine side was brought up to about .040 kg/s, about halfway to the final operating point. The idea was to start the combustion at this lower mass flow rate of 0.040 kg/s to ensure sustainable combustion. After ignition the mass flow through the combustor was walked up to the final 0.071 kg/s operating point.

When the air reached 0.040 kg/s, the ignition process was started. As can be seen in Figure 48 and 49 this happened at roughly 50 seconds into the test. At this point the compressor-turbine couple quickly sped up, which was clearly audible as a large whining sound and evident in the results. At the point of ignition the results showed that the compressor was outputting roughly 0.035 kg/s while the turbine was outputting 0.04 kg/s. Even though matching was not achieved, the combustion process brought the mass flows much closer. Even though small, there was an increase in pressure on the compressor side that correlated with the introduction of combustion.

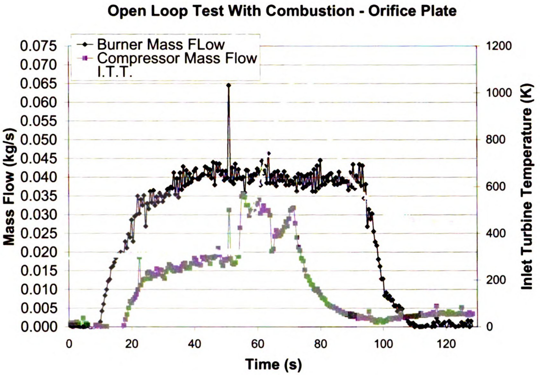

Figure 48. Open-loop turbocharger test with combustion, mass flow and I.T.T.

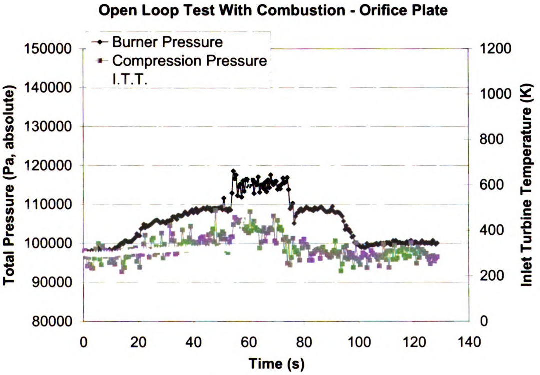

Figure 49. Open-loop turbocharger test with combustion, pressure and I.T.T

Unfortunately, the compressor only sped up for a short while before the loud popping sound was witnessed again. This time the popping was followed by a sudden lock of the compressor. Once it was noticed that the compressor was no longer spinning even though there was still flow through the combustor, the system was quickly shut down. After disassembly it was apparent that the compressor had slammed forward against the volute. This clearly supported the idea that the compressor reached its limit and surged. Fortunately, this surge did not break the shaft between the compressor and turbine. Removing the volute on the front of the turbocharger allowed for the compressor to be freed. The compressor blades were slightly damaged by coming in contact with the volute. This damage to the blades will affect the performance of the compressor and will most likely increase the probability of surging again.

Due to the surge, the operating point of 0.071 kg/s was never achieved. Even though surging was again likely another test was run to determine if the compressor truly reached its surge limit or if the previous test was a one-time irregularity.

The second test was run in the same manner (Figure 50, 51). Like in the previous test the introduction of combustion caused the compressor to start popping loudly. Only this time the compressor did not lock into the volute. It continued to pop once or twice every few seconds. This popping took place for roughly 20 seconds until the mass flow in the compressor side began to drop rapidly. Correspondingly, the flow through the turbine began to rise. This would suggest that the shaft had broken. After the test was shut down, the turbocharger was again disassembled and it was confirmed that the compressor had surged so violently that the shaft broke.

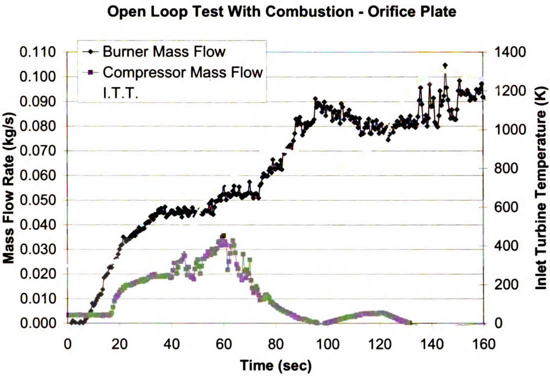

Figure 50. Open-loop turbocharger test with combustion, mass flow and I.T.T.

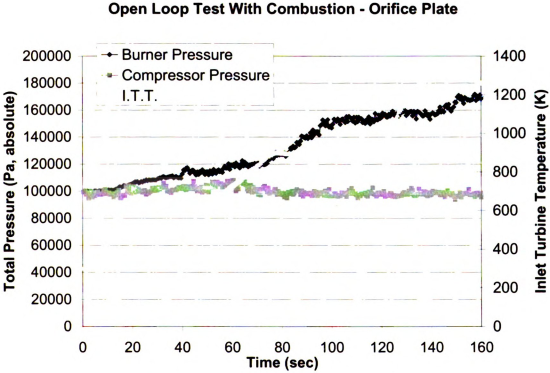

Figure 51. Open-loop turbocharger test with combustion, pressure and I.T.T.

WAVE ROTOR TESTING

The other half of the combustion chamber and test rig design was for the evaluation of the wave rotor. The wave rotor was tested in the same open-loop configuration as the turbocharger, meaning there was a separation of the compression and expansion flows of the system (Figure 52). Like in the turbocharger tests the combustion chamber was routed upstream of the expansion side of the wave rotor. The compression inlet was allowed to pull air freely from ambient. The hot combustion gas expanded in the wave rotor driving the compression process. Fresh air was pulled in and compressed. This compressed air was ejected from the wave rotor.

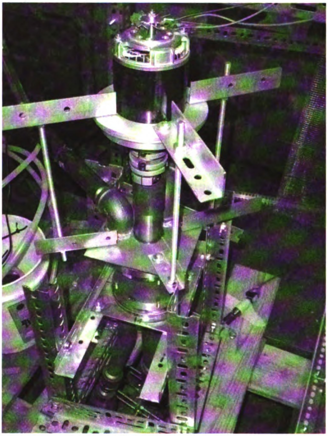

Figure 52. Wave rotor test apparatus

The first step was to light the pilot light. Once the pilot light was lit, the wave rotor was sped up to 13,000 rpm by an external electrical motor. The high pressure line was again used to force air through the combustor and to the expansion ports of the wave rotor. The air in the combustor was first brought up to the desired mass flow rate. At this point the fuel was added. As expected, there was a low thud audible and combustion was apparent. Being the first ever test run with the wave rotor, it was determined to first operate the wave rotor with combustion gases no higher than 900 Kelvin. This operating point was sustained for roughly 30 seconds and then the test was systematically shut down. Fuel was removed and then the wave rotor was gradually returned to zero rpm while the air flow through the combustor was being shut off. When safe to approach the rig, the last step was to turn the pilot off.

The types of parameters acquired for the wave rotor are identical to previous tests. The pressure, temperature, and mass flow of the expansion side were acquired as well as the pressure, temperature, and mass flow of the compression side. In the results below, the mass flow through the expansion side was much higher than the mass flow through the compression side, roughly 8 times as much. When at the proper operating condition, the mass flows should be roughly equal. Data suggest the point at which this particular test was conducted was not the optimal one (Figure 53, 54). However, the wave rotor did show that it was capable of ingesting and compressing some amount of ambient air.

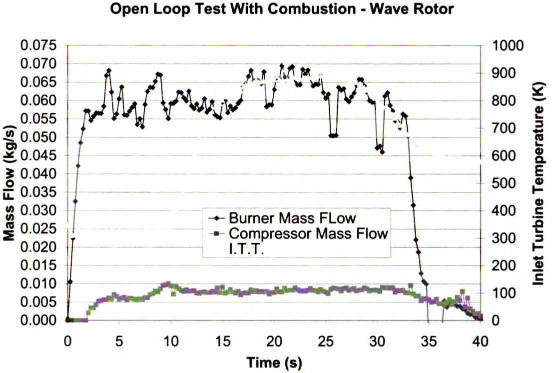

Figure 53. Open-loop wave rotor test with combustion, mass flow and I.T.T.

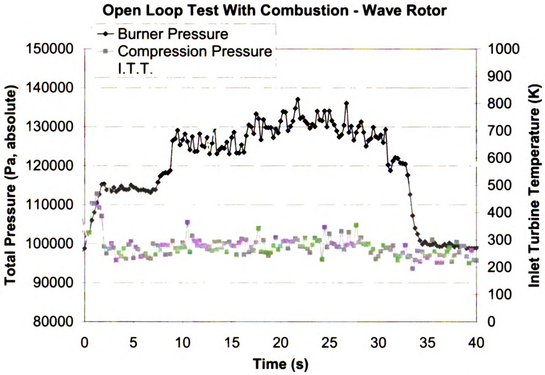

Figure 54. Open-loop wave rotor test with combustion, pressure and I.T.T.

The test rig proved capable of running the wave rotor at high speeds and delivering the combustion gases needed to evaluate the technology, which was one of the primary goals of this test rig. With the test rig procedure set, the next step for the up coming generation of tests is to determine a series of operating ranges and conditions to explore and optimize the capabilities of the wave rotor technology.

CONCLUSIONS

In design and implementation, the test rig is considered a success. The combustion chamber proved to be capable of the operating ranges demanded by the turbocharger and wave rotor test plans. Decelerating the combustion air over a larger cross sectional area, introducing a continuous pilot flame, and introducing the central flame holder proved to be crucial aspects to the success of the combustion chamber design.

In the closed-loop configuration the turbocharger was not capable of accelerating up to the speed and mass flow necessary to operate the test rig as a 5 kW gas turbine engine. An open-loop test showed that the engine did accelerate in the presence of combustion. However, this acceleration did not carry the engine up to the desired operating point. Before the operating point could be achieved the compressor repeatedly surged causing a sudden deceleration and the eventual destruction of the compressor – turbine shaft. This turbocharger ran close to its surge line at the operating point. Another type of turbocharger where the operating point could be placed further away from the surge line would be more desirable.

The wave rotor testing configuration has been established. This preliminary testing proved that the test rig and procedure were capable of providing the wide flow rates and combustion temperatures needed to test the wave rotor. Several test runs have been conducted at this point. The next step is to run the wave rotor at various operating conditions and map the results. These results are crucial for understanding and validating the performance enhancing ability of the wave rotor technology by determining the operating conditions at which the benefits exist. The rig and testing procedures have been set. This next phase of testing can now be conducted.

RECOMMENDATIONS

There are several reasons that the current turbocharger was experiencing compressor surge before achieving the operating point. The first was that the operating point chosen was very closely to the surge limit for this chosen turbocharger. This surge limit was defined by the compressor map, which comes as product data that was produced by testing from the turbocharger manufacturer. One reason the turbocharger may be experiencing surge at a lower operating point than indicated by the factory data could be due to inlet distortion. This means that the incoming flow to the compressor was probably much straighter and conditioned by a special shaped inlet bell mouth, or other geometry, when the compressor map data was generated by the manufacturer. The current test rig does not utilize any type of inlet geometry to limit inlet distortion other than the volute itself. For example, incorporating an A.S.M.E. bell mouth design would drastically improve the incoming flow of air to the compressor and decrease the likelihood of surging the compressor [A.S.M.E., 2007]. The A.S.M.E. standard bell mouth shape is widely proven and used in the aerospace industry. The specific geometry of the bell mouth is a simple elliptical shape. The exact dimensions of the inlet are defined characteristically by the diameter of the inlet within the A.S.M.E standard. It is highly recommended that this bell mouth be manufactured and implemented for any further engine testing. A view of what an A.S.M.E. standard bell mouth would look like for the current Garrett turbocharger being used can be seen in Figure 55. If it is determined that a different turbocharger will be used for subsequent testing, then the exact dimensions will simply change proportionally relative to the increase or decrease of the inlet diameter.

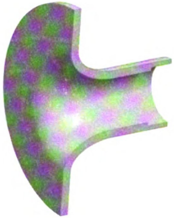

Figure 55. ASME bell mouth scaled for use with gas turbine engine, cross section view

During testing the first and last step was always to manually light and then extinguish the pilot light for the combustor. It would be an advantage to be able to automate this process because lighting the pilot requires coming in contact with the test rig. The ultimate goal for this test rig is to be run from one central computer station safely away from the rig itself. Therefore a device that could replace the current MAPP gas pilot light that could be controlled electronically from a remote location would be best. A device that could achieve this is a hot surface igniter. This igniter is commonly used for igniting residential furnaces for home heating. The igniter is a simple heating element that acts as an electrical resistor and heats up when energized with standard 110 Volt AC power. There are several variations of this type of product on the market.

Figure 56 below shows a purchased igniter with the element energized. This igniter is already located in the lab and simply needs to be wired and mounted within the combustion chamber and routed to the computer control station with an appropriate switch for control. This igniter has been tested and proven to be capable of igniting the propane.

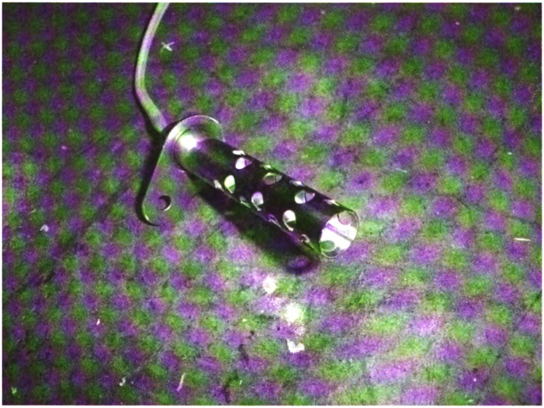

Figure 56. Hot surface igniter commonly used in residential furnaces

Another step towards automating the test rig would be to incorporate a fuel control system that could be controlled from the command computer. The current fuel control meter can be computer controlled but was found to not have the capacity to deliver the mass flow rate of fuel needed. A flow meter, similar in technology to the current meter, should be obtained but capable of a larger mass flow rate should be obtained.

Also, the oil pump that feeds oil to the bearings for lubrication and cooling could be automated. A simple electric motor with a potentiometer for speed control could be coupled to the shaft of the oil pump. Currently, the oil pump is driven by an AC drill and the speed can not be altered with out first coming in contact with the test rig. A new motor with a potentiometer could be wired to the control station and modulated from there.

During testing it was noticed that there were some small air leaks around the flanges of the combustion chamber. To prevent this leaking the flanges should be sealed with some type of gasket material. The combustion chamber is capable of reaching 1200 K near the outer wall; therefore any gasket chosen must be capable of this.

The last modification recommended would be to install a ventilation system in the lab capable of removing the exhaust gas from the lab area and expelling it outside. Currently, the exhaust gas is routed out of the test lab through metal ducting. However, it is inevitable that some of the exhaust gas does not completely leave the testing area and this ventilation system should be capable of circulating these gases out.

For the wave rotor the ground work for testing has been set and future testing with this rig can now be conducted to further develop and understand this novel technology.

WORKS CITED

1. Akbari, P, Nalim, R, Mueller, N. "Performance Enhancement of Microturbine Engines Topped With Wave Rotors." Journal of Engineering for Gas Turbines and Power. 128 (2006): 190-202.

2. ASME MFC-3Ma-2007, "Measurement of Fluid Flow in Pipes Using Orifice, Nozzle, and Venturi."

3. Dempsey, E, Mueller, N, Akbari, P, Nalim, M R. "Performance Optimization of Gas Turbines Utilizing Four-Port Wave Rotors." AIAA Pap. 2006-4152.

4. Garrett. Garrett Product Catalog. <http://www.turbobygarret.com/turbobygarrett/catelog/turbochargers/gt12/gt1241_756068_1.htm>

5. Kusner, M, "Design of a 5 kW Microturbine Generator." East Lansing, MI: Michigan Stat University, 2006.

BIBLIOGRAPHY

1. Akbari, P, and Szpynda, E, Nalim, M R. "Recent Developments in Wave Rotor Combustion Technology and Future Perspectives: A Progress Review." AIAA2007-5055.

2. Beer, F P, Johnston Jr, E R, DeWolf, J T. *Mechanics of Materials*. 3rd ed. New York: McGraw-Hill, 2001.

3. Cumpsty, N. *Jet Propulsion*. Cambridge: Cambridge University Press, 1997.

4. Davison, C, and Birk, A M. "Set Up and Operational Experience with a Micro-Turbine Engine for Research and Education." Proceedings of ASME Turbo Expo 2004, GT2004-53377.

5. Dixon, S L. *Fluid Mechanics, Thermodynamics of Turbomachinery*. 4th ed. Massachusetts: Butterworth-Heinemann, 1998.

6. Incropera, F P, DeWitt, D P. *Introduction to Heat Transfer*. 4th ed. New York: Wiley, 2002.

7. Kuo, K. *Principles of Combustion*. New Jersey: Wiley, 2005.

8. Moran, M J, Shapiro, H N. *Fundamentals of Engineering Thermodynamics*. 5th ed. New Jersey: Wiley, 2004.

9. Mueller, N and Frechette, L. "Performance Analysis of Brayton and Rankine Cycle Microsystems for Portable Power Generation." ASME International Mechanical Engineering Congress & Exposition. New Orleans, LA, 1-10.

10. Nijeholt, J, Komen, E, Verhage, A, van Beek, M. "First Assessment of Biogas Co-Firing on the GE MS9001FA Gas Turbine Using CFD." Proceedings of ASME Turbo Expo 2003, GT2003-38804.

11. Potter, M, Wiggert, D. *Mechanics of Fluids*. 3rd ed. California: Brooks/Cole, 2002.

12. Treager, I. *Aircraft Gas Turbine Engine Technology*. 2nd ed. New York: McGraw-Hill, 1979.

13. Tsai, L. "Design and Performance of a Gas-Turbine Engine from and Automobile Turbocharger." Cambridge, MA: Massachusetts Institute of Technology, 2004.

MICHIGAN STATE UNIVERSITY LIBRARIES
3 1293 02956 8585