Chapter 1

Introduction

The ever present desire for increased efficiency and reduced emissions from gas turbine combustors is addressed in this thesis through the development of a facility and methodology for the testing of a hydrogen-air mixer. During this study, three experiments are implemented to evaluate different mixer performance characteristics: Mie scattering flow visualization, schlieren flow visualization, and Laser Doppler Velocimetry (LDV). Mixer performance is evaluated at and downstream of the mixer outlet by measuring: relative fuel concentration uniformity and spatial distribution, overall swirl angle, and the 3-D velocity profile. This work is primarily motivated by the benefits of hydrogen combustion in gas turbine engines, such as increased efficiency, reduced emissions and fuel flexibility. A methodology for overlapping images is also developed to allow experimental results to be compared with CFD simulation results for validation purposes. Results from both flow visualization studies are presented to illustrate operation of the facility and methodology developed. This section outlines the motivations and objectives of the study, and also provides background information along with a discussion of the technical approach followed during the course of this study.

1.1 Motivation and Objectives

At the highest level, this study is motivated by environmental interest in the combustion of alternative fuels in gas turbine engines and the political interest in moving the United States towards the adoption of a hydrogen economy. As Environmental Protection Agency (EPA) regulated emission limits for gas turbines continue to become more stringent, the gas turbine industry is constantly researching new methods for reducing emissions. Three of the most important pollutant gases are Carbon Monoxide (CO), Carbon Dioxide (CO_2), and Nitrous Oxides (NO_x). The most advanced combustor designs for traditional fuels, such as Kerosene and Diesel, are rapidly reaching the limit of how far their emissions can be reduced. Combustors designed to burn new fuels such as hydrogen offer the potential to continue the reduction of pollutant emissions beyond current technology. Hydrogen combustion completely eliminates the source of carbon from the combustion reaction and as a result CO and CO_2 emissions are eliminated.

Hydrogen has a higher flame temperature than traditional fuels and therefore has the potential to produce higher NO_x emissions. However, Brand et. al. [1] have shown that hydrogen, which has wider flammability limits then traditional fuels, can be burned at leaner equivalence ratios and as such, NO_x production can be reduced. The higher flame speed related to hydrogen-air combustion, reduces residence time in the combustion region which also contributes to reduced NO_x formation.

As mentioned previously, the environmental issues and high cost of fossil fuels are motivating strong political interest in the use of hydrogen. An example of hydrogen legislation from the House of Representatives is H.R. 5243, which "establishes competitively awarded cash prizes for scientific breakthroughs in the advancement of hydrogen energy technology [2]." The US Department of Energy (DOE) is also creating initiatives to increase and facilitate academic research into Hydrogen combustion.

One of the major design changes when modifying a gas turbine engine (operating on traditional fuels) for operation on hydrogen is the design of the fuel-air mixer. A mixer must be designed specifically for mixing hydrogen and air to achieve the maximum efficiency and minimum emissions. In addition, a facility and methodology for the analysis of such a mixer must be established.

The main objective of the study presented here is to establish a facility for hydrogen-air mixer evaluation. To this end, three experiments are implemented to test several performance characteristics of a mixer, namely; Mie scattering flow visualization, schlieren flow visualization, and LDV. Each method provides insight into different performance characteristics, which include: relative fuel concentration uniformity and spatial distribution, overall swirl angle, and velocity profile.

Due to the expense of, and time consumed during physical nozzle testing, Computational Fluid Dynamics (CFD) is a more economical design tool. However, CFD results for new mixer geometries are only estimates, and must be validated by comparison with the results of physical tests before use. The facility developed through this study provides the capability to perform physical tests for validation. As a secondary objective of this study, a methodology is developed for the comparison of CFD and experimental results. The final objective of this thesis is to illustrate the utility of the facility and methodology developed during this study through the evaluation of two hydrogen-air mixing nozzle designs. The methodology discussed is then used to compare the experimental results to CFD simulations.

The nozzles evaluated were designed by David Sykes and Joseph Homitz, and are referred to as Nozzle D and Nozzle J respectively. These students also generated the CFD models for each nozzle. The specific CFD results provided in this thesis were generated by the author.

1.2 Background and Technical Approach

This section provides fundamental technical background on the three experimental techniques implemented in this study: Mie scattering flow visualization, schlieren flow visualization, and LDV. Following the background information, the tested operating conditions and the testing plan are discussed.

1.2.1 Mie Scattering Flow Visualization

Throughout the history of aerodynamic study, researchers have desired to make complex flow patterns visible. The ability to visualize a flow improves our understanding of the major physical phenomenon that cause the flow to act as it does. Flow visualization studies are commonly used to provide qualitative insight into a specific flow pattern; however, experiments can be designed to also provide quantitative results. One of the major advantages offered by the flow visualization technique is that it is nonintrusive, there are no probes introduced to the flow causing disruption. The clean air delivered from the main air supply is transparent but can be seen when smoke, or other particulate matter is introduced and illuminated by a sufficiently intense light source. Care must be taken to ensure that a particle size is selected which will allow the particles to properly follow the true nature of the flow being observed. For the particles to follow the flow, they must have a small size and thus, low inertia. The smoke particles selected must also be large enough to scatter sufficient light in order to be clearly photographed. Many different materials may be burned or vaporized to generate smoke. Vaporized motor oil is used in the present study. The oil particle sizes range from 0.30 to 1.0μm [3].

Once the flow has been injected with the properly selected seeding, a light source must be provided to illuminate the smoke or 'seed' particles. Typically, a high intensity source such as a laser, arc lamp, or strobe light is used. The higher the intensity of the source, the easier it becomes to acquire a clear photograph of the flow. In this experiment, a laser sheet implementation is used to illuminate the z-y plane of the mixer outlet flow. A pair of cylindrical lenses is a key component of the laser sheet optics responsible for spreading a laser beam into a sheet. The sheet optics include a final mirror mounted to an x-axis translation stage, which allows the sheet to be scanned across the flow to cut any cross section desired.

1.2.2 Schlieren Flow Visualization

The schlieren flow visualization technique reveals density gradients across a flow field. This technique may also be called an index-of-refraction technique because variations in the index of refraction, caused by density gradients within the flow, are measured directly. In a schlieren system, the image captured represents the first derivative of the index of refraction in a direction normal to the illuminating beam of light. Using the coordinate system of this study, illustrated in Figure 1-1, the first derivative of the index of refraction is described by the following equation,

$$\frac{\partial \rho}{\partial y} = \frac{1}{C}\frac{\partial n}{\partial y} = \frac{\rho_O}{n_O - 1}\frac{\partial n}{\partial y} \tag{1-1}$$

where ρ is the fluid density, n is the index of refraction, ρ_O and n_O are at standard atmospheric conditions, and C is the Gladstone-Dale constant, which is a function of the particular gas [3].

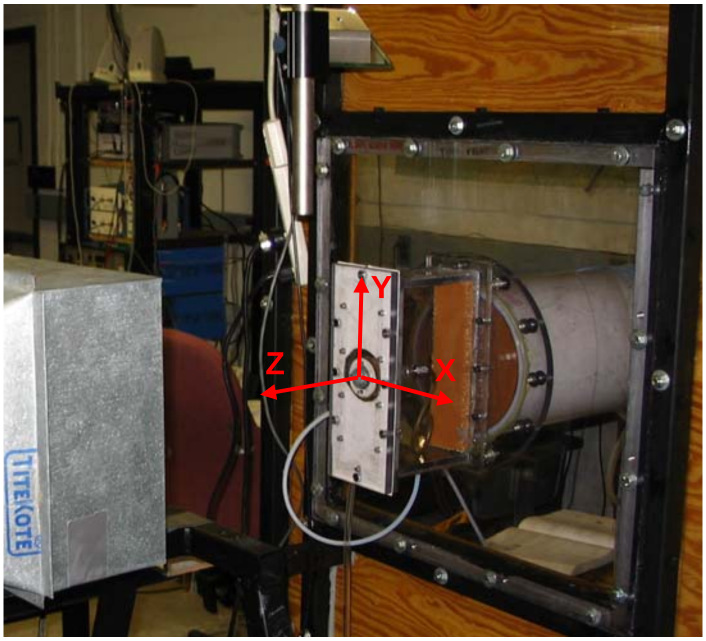

Figure 1-1. Global coordinate system for this study with the origin on the center of the nozzle outlet.

The first optical components of a schlieren system include a light source and lens, which establish a collimated light beam passing through the test section. The light beam then passes through another lens, which focuses the collimated beam to a knife edge. The knife edge darkens the image which is finally captured by a digital camera. In the test section across a boundary between two areas of differing density, there is a change in the index of refraction of the gas. The light passes through the test section and is altered by changes in the index of refraction, which can be readily seen. Figure 1-2 shows a traditional schlieren system.

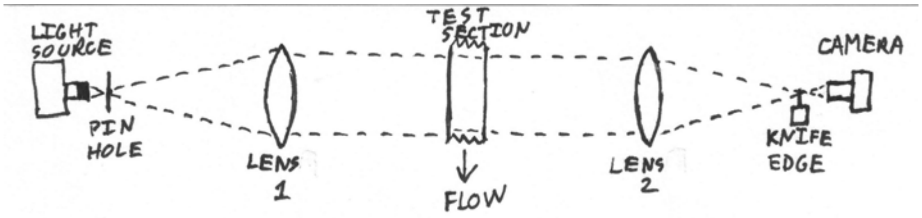

Figure 1-2. Traditional schlieren system using lenses to generate a collimated light beam.

As with the Mie scattering flow visualization, schlieren flow visualization has the advantage of being a nonintrusive measurement technique. Schlieren does however, require high quality plane windows for optical access to the test section. This was discovered when Plexiglas windows were initially installed resulting in non useful images. The schlieren system integrates the quantity being measured, in this case density, over the length of the light beam. Thus, the schlieren technique measures an average density along the collimated light beam passing through a 3-D flow.

1.2.3 Laser Doppler Velocimetry

LDV is a flow measurement technique in which the fluid velocity is measured through detection of the Doppler frequency shift of laser light scattered from small particles moving with the flow. As with the previously discussed visualization studies, LDV has the advantage of being a nonintrusive fluid measurement technique useful for measuring many different types of flows. LDV measures light scattered from small particles, typically 0.1 to 10 μm [3]. These particles are small enough to accurately follow the fluid in which they are immersed, and large enough to scatter sufficient light to be detected by an opto-electric sensor such as a Photo Multiplier Tube (PMT).

Throughout the last three decades, LDV has been a popular technique for the measurement of flow velocities [3]. There are primarily two models that can be used to explain the operating principles of the LDV, the fringe model and the Doppler shift model. The former is easier to understand and is discussed here. In this study, a dual beam backscatter LDV system is implemented. To measure a single velocity component, two light beams of equal intensity and wavelength, and different angle, intersect in a location called the probe volume. In the ellipsoidal probe volume, the waves of the two beams interfere and create a set of interference fringes, or diffraction pattern, as shown in Figure 1-3. Because the illuminating beams have equal frequency, a standing interference pattern is produced. The illuminating beams are round and have a Gaussian intensity distribution.

Equation 1-2 mathematically describes the sum of the intensity of the two beams,

$$I_0(x) = I_1(x) + I_2(x) + 2\sqrt{I_1(x)I_2(x)}\cos(\omega_S t + 2kx\sin(\kappa)) \tag{1-2}$$

I_1 and I_2 are the intensities of the illuminating beams, ω is the frequency of the light in rad/s, t is time in seconds, k is wave number and κ is the half angle between the illuminating beams. The third term in Equation 1-2 represents the interference fringes. The spacing between fringes is given by Equation 1-3,

$$2kd_f \sin(\kappa) = 2\pi \tag{1-3}$$

Substituting in the definition of wave number, $k=2\pi/\lambda$, provides,

$$d_f = \frac{\lambda}{2\sin(\kappa)} \tag{1-4}$$

The scattered light flux of the particle as it moves across the fringes in the probe volume will oscillate sinusoidally with a frequency of,

$$f = \frac{u}{d_f} = \frac{2u\sin(\kappa)}{\lambda} \tag{1-5}$$

the Doppler frequency, where u is the particle velocity in m/s and λ is the wavelength of the illuminating beams. If the Doppler frequency of a passing particle is measured, Equation 1-5 may be used to compute the particle velocity.

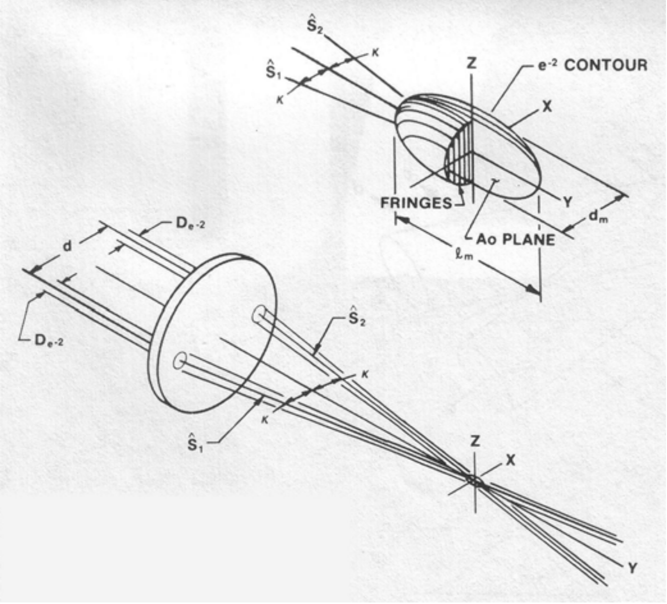

Figure 1-3. Illustration of the probe volume of a Dual Beam LDV system. The beam pair shown can be used to measure the velocity component along the x axis [4].

This LDV implementation is called a backscatter system because the light scattered from the probe volume is collected in the direction from which the illuminating beams came, as shown in Figure 1-4. Another common LDV implementation is the forward scatter system, where the illuminating beams enter one side of the test section and the scattered light is collected on the opposite side, along the forward direction of the beams. In either case, the receiving optics capture the scattered light and deliver it to a PMT which converts the light flux into current. An op-amp circuit or load resistor can be used to convert the PMT current output into a voltage output which is less challenging to measure.

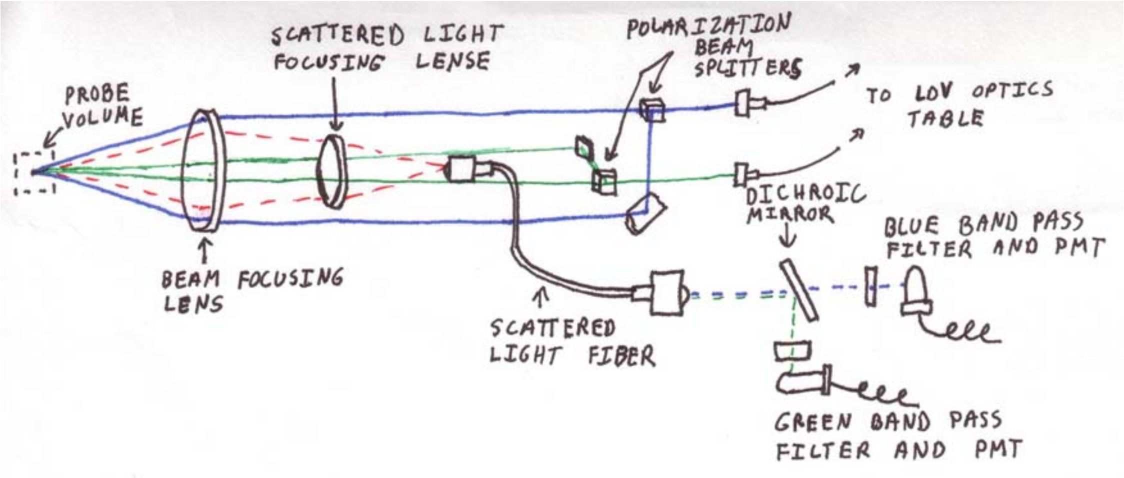

Figure 1-4. Illustration of two component backscatter LDV system implemented in this study.

The two component LDV implemented in this study uses two beam pairs of different color to measure two velocity components. A green (514nm wavelength) pair measures the axial velocity component and a blue (488nm wavelength) pair measures the tangential or swirl velocity component. These beam pairs meet orthogonally in the probe volume, allowing the measurement of orthogonal velocity components.

1.3 Scope

The primary goal of this study is to develop the three experiments discussed previously, as a facility for the evaluation of gaseous fuel-air mixers. Both flow visualization techniques are fully operational and the LDV is functioning in a self test configuration. Detailed facility notes and procedures are included in this document so that the facility developed may be easily operated and maintained by future Virginia Tech students for new projects.

To illustrate the effectiveness of the facility developed, two mixer nozzles are tested using the Mie scattering and schlieren flow visualization experiments. Both experiments focus on analysis of the nozzle outlet flow. For the Mie scattering experiments, three equivalence ratios (as defined in equation 1-6) are tested: 0.3, 0.4, and 0.5, each at the on-design air flow rate of 0.0199 kg/s.

$$\phi = \frac{\left(m_{fuel}/m_{air}\right)_{actual}}{\left(m_{fuel}/m_{air}\right)_{stoichiometric}} \tag{1-6}$$

Where Phi is equivalence ratio, and the 'm' terms are masses. Schlieren experiments for each nozzle are conducted at the on-design air flow with an equivalence ratio of 0.4. Both experiments provide solely qualitatively insight into several characteristics of the fuel air mixing, geometric features of the flow, and features of the swirl. If advanced into a nozzle testing configuration, the LDV can provide quantitative insight into mixer performance. When mounted on the three axis translation stage with control code already developed, the LDV can automatically map the entire 3-D nozzle outlet flow velocity field at any resolution up to 1mm. As an added facility capability, the static pressure drop across a nozzle can also be measured.

The methodology developed for comparison of experimental and computational results is illustrated through the comparison of the nozzle pressure drop, Mie scattering and schlieren flow visualization results to CFD results for the two nozzles evaluated. These comparisons are then used to qualitatively validate the computational predictions.

1.4 Thesis Structure

The following sections of this document describe in detail, the technical approach used to develop the results presented. The test facility and measurements made are discussed along with a presentation of the results of each test. The experimental results are then compared with the CFD results and discussed. This is followed by the main conclusions. Finally, recommendations are made for ways the test facility and procedures can be upgraded in the future to provide more useful and higher quality results. The Appendices contain additional information about: testing procedures and equipment maintenance for the three experiments developed, rig air supply system and compressors, and translation stage development, operation, and maintenance.