Heat transfer and flow characteristics inside a gas turbine combustor

Yap-Sheng Goh
Louisiana State University and Agricultural and Mechanical College, ygoh1@lsu.edu

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_theses

Part of the Mechanical Engineering Commons

Recommended Citation
https://digitalcommons.lsu.edu/gradschool_theses/3832

This Thesis is brought to you for free and open access by the Graduate School at LSU Digital Commons. It has been accepted for inclusion in LSU Master's Theses by an authorized graduate school editor of LSU Digital Commons. For more information, please contact gradetd@lsu.edu.
HEAT TRANSFER AND FLOW CHARACTERISTICS INSIDE A GAS TURBINE COMBUSTOR

A Thesis

Submitted to the Graduate Faculty of the Louisiana State University and Agricultural and Mechanical College in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering

in

The Department of Mechanical Engineering

by

Yap-Sheng Goh
B.S., Louisiana State University, May 2003 December 2006
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES</td>
<td>iii</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>vi</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>viii</td>
</tr>
<tr>
<td>CHAPTER 1: INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Cooling Systems</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Pollutant Emissions</td>
<td>3</td>
</tr>
<tr>
<td>1.3 Water/Steam Injection</td>
<td>7</td>
</tr>
<tr>
<td>1.4 Dry Low Emission (DLE) Combustor</td>
<td>7</td>
</tr>
<tr>
<td>1.5 Focused Cooling</td>
<td>10</td>
</tr>
<tr>
<td>1.6 Swirler</td>
<td>10</td>
</tr>
<tr>
<td>1.7 Literature Survey</td>
<td>13</td>
</tr>
<tr>
<td>1.8 Experimental Objectives</td>
<td>14</td>
</tr>
<tr>
<td>CHAPTER 2: EXPERIMENTED APPARATUS</td>
<td>16</td>
</tr>
<tr>
<td>2.1 Air Supply</td>
<td>16</td>
</tr>
<tr>
<td>2.2 Inlet Chamber and Swirler</td>
<td>19</td>
</tr>
<tr>
<td>2.3 Main Combustion Chamber</td>
<td>23</td>
</tr>
<tr>
<td>2.4 Exit Dump Diffuser</td>
<td>25</td>
</tr>
<tr>
<td>2.5 Heat Transfer Experiment</td>
<td>25</td>
</tr>
<tr>
<td>2.5.1 FLIR SC500 Infrared Thermal Imaging System</td>
<td>25</td>
</tr>
<tr>
<td>2.5.2 INSTRUNET Temperature Measurement System</td>
<td>27</td>
</tr>
<tr>
<td>2.5.3 Wall Surface Heaters</td>
<td>28</td>
</tr>
<tr>
<td>2.6 Flow Characteristic Experiment</td>
<td>33</td>
</tr>
<tr>
<td>2.6.1 TSI IFA300 Hot-wire Constant Temperature Anemometry</td>
<td>33</td>
</tr>
<tr>
<td>CHAPTER 3: EXPERIMENTED METHODOLOGY</td>
<td>39</td>
</tr>
<tr>
<td>3.1 Steady-State Temperature and Heat Transfer Measurements</td>
<td>41</td>
</tr>
<tr>
<td>3.2 Hot-wire CTA Turbulence and Velocity Flow Field Measurements</td>
<td>42</td>
</tr>
<tr>
<td>3.3 Integral Length Scale Analysis</td>
<td>44</td>
</tr>
<tr>
<td>CHAPTER 4: RESULTS</td>
<td>48</td>
</tr>
<tr>
<td>4.1 Flow Data</td>
<td>48</td>
</tr>
<tr>
<td>4.2 Heat Transfer Data</td>
<td>55</td>
</tr>
<tr>
<td>4.3 Comparison of Heat Transfer and Flow Results</td>
<td>64</td>
</tr>
<tr>
<td>CHAPTER 5: CONCLUSIONS</td>
<td>66</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>69</td>
</tr>
<tr>
<td>VITA</td>
<td>72</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 1.1: Typical Combustor Liner Wall Film Cooling .................................................. 2
Figure 1.2: Influence of Primary-Zone Temperature on CO and NOx emissions .......... 5
Figure 1.3: Solar Turbines MARS Low-NOx Combustor ................................................. 9
Figure 2.1: Overview of Experimental Setup .................................................................... 16
Figure 2.2: AC TECH Variable Speed AC Motor High Capacity Fan .............................. 18
Figure 2.3: Fan Performance Curve .................................................................................. 19
Figure 2.4: Combustion and Inlet Chambers with Swirler ............................................. 20
Figure 2.5: Solar Turbines Swirler .................................................................................... 21
Figure 2.6: 3-D Solid Model of Solar Turbines Swirler ................................................... 21
Figure 2.7: Schematic of Swirler ...................................................................................... 22
Figure 2.8: IR Camera Windows Cutoff Locations ......................................................... 23
Figure 2.9: Hotwire Probe Mounting Rail ........................................................................ 24
Figure 2.10: Exit Dump Diffuser ..................................................................................... 25
Figure 2.11: FLIR SC500 Infrared Camera ...................................................................... 26
Figure 2.12: InstruNet Sensor Data Acquisition Unit ....................................................... 28
Figure 2.13: Wall Surface Heater System ....................................................................... 29
Figure 2.14: Construction Layers of Heater System ....................................................... 29
Figure 2.15: Stainless Steel Shim Stock ......................................................................... 30
Figure 2.16: Hi-temp Heat Resistant Flat Black Paint ...................................................... 32
Figure 2.17: Conductive Heat Loss .................................................................................. 32
Figure 2.18: 120-volt Transformer ................................................................................... 33
Figure 2.19: Hotwire Constant Temperature Anemometry System ................................. 35
Figure 2.20: TSI 1128B Hotwire Air Velocity Calibrator .............................................. 35
Figure 2.21: Overview of TSI IFA 300 CTA Calibration System .................................... 37
Figure 2.22: Hotwire Calibration Curve ........................................................................... 38
Figure 3.1: General Idea of Flow Behavior ...................................................................... 39
Figure 3.2: IFA 300 Constant Temperature Anemometry Unit ....................................... 43
Figure 3.3: Film Type Dual Sensor X-Probe ................................................................. 43
Figure 3.4: Turbulent Flow Velocity Profiles ................................................................. 44
Figure 4.1: U-component Turbulence Intensity Distribution ........................................... 48
Figure 4.2: Turbulence Intensity Distribution ................................................................. 49
Figure 4.3: U-component Velocity Flow Field ............................................................... 50
Figure 4.4: Total Velocity Flow Field ............................................................................... 51
Figure 4.5: Turbulence Integral Length Scale ................................................................. 53
Figure 4.6: Turbulence Integral Time Scale ..................................................................... 54
Figure 4.7: Spanwise Averaged Turbulence Integral Length Scale ............................... 54
Figure 4.8: 2-D Temperature Distribution .................................................................... 55
Figure 4.9: 2-D Heat Transfer Coefficient Distribution ............................................... 56
Figure 4.10: 1-D Heat Transfer Distribution .................................................................. 58
Figure 4.11: Swirling Effects on Averaged Heat Transfer Distribution ......................... 60
Figure 4.12: Comparison of Heat Transfer Coefficient Distribution ............................ 61
Figure 4.13: Comparison of Local Nusselt Number Distribution with respect to 
Combustor diameter for Re=500000 and 662000 ...................................................... 61
Figure 4.14: Comparison of Local Nusselt Number Distribution with respect to axial 
distance for Re=500000 and 662000 ........................................................................ 62
Figure 4.15: Comparison of Local Nusselt Number Distribution to FDF Straight-pipe Estimation for Re=500000 and 662000 ................................................................. 63

Figure 4.16: The overlapping of Heat Transfer and Turbulence Intensity Distribution for Re=500,000 and Re=662,000 ........................................................................ 65
NOMENCLATURE

\( D \) diameter of main combustion chamber
\( D_h \) hydraulic diameter of swirler
\( D_i \) inner diameter of swirler
\( D_o \) outer diameter of swirler
\( D_{sw} \) mean diameter of swirler
\( G_m \) axial flux of angular momentum
\( G_t \) axial trust
\( A \) effective cross-sectional area of swirler
\( A_m \) cross-sectional area of main combustion chamber
\( A_{ss} \) cross-sectional area of stainless steel foil
\( p \) effective perimeter of swirler
\( \gamma \) ratio of the specific heat at constant pressure to the specific heat at constant volume
\( R_g \) universal gas constant
\( R \) electrical resistance
\( R(t) \) autocorrelation function
\( L \) turbulence integral length scale
\( J \) turbulence integral time scale
\( Q \) volumetric flow rate
\( \overline{Q} \) heat generation
\( I \) electrical current
\( V \) electrical voltage
\( P_E \) electrical power output
\( a \) local speed of sound
\( v \) air velocity
\( \rho \) electrical resistivity
$L_e$ effective length

$h$ local convection heat transfer coefficient

$k$ thermal conductivity

$\alpha$ thermal diffusivity

$\nu$ kinematic viscosity

$Nu$ Nusselt number

Re Reynolds number

$M$ Mach number

$U$ average air velocity

$S_N$ swirl number

$T$ air temperature

$r$ flow expansion ratio
ABSTRACT

Heat transfer and flow characteristics inside a typical can-annulus gas turbine combustor are investigated. This is the first study in a public domain to focus on the convective heat loads to combustor liner due to swirling flow generated by swirler nozzles. The objectives of this study are to physically design an experimental combustor test model, to perform local accurate heat transfer and flow measurements, and to better understand the fundamental thermo-fluid dynamic effects inside a combustor equipped with a swirler nozzle provided by Solar Turbines Inc. The local temperature and heat transfer distribution were determined to locate the heat transfer peak region and compared with velocity flow field and turbulent intensity distributions.

The actual test model is 5-times the original 8” (20.32cm) diameter prototype to simulate realistic engine operating conditions and also provide high resolution measurements. Experiments were performed at two flow Reynolds Numbers (500k and 662k) to further investigate the effects of Reynolds Number on the heat transfer peak locations and velocity flow field distributions.

The heat transfer investigation was performed using the steady-state Infrared Thermography technique. Six identical surface heater foils were used to simulate the constant-heat-flux boundary condition, and the Infrared Thermal Imaging system was used to capture the real-time steady-state temperature distribution at the combustor liner wall. The results show that the heat transfer peak regions at different Reynolds numbers occur at the same exact location.

Investigations on flow characteristics were also performed to compare the velocity flow field and turbulent intensity distributions with the heat transfer results.
Using TSI Constant Temperature Anemometry technique, with a highly sensitive hot-wire dual sensor X-probe and a rigidity reinforced probe support system; the 3-dimensional complex swirling velocity flow field was measured.

The heat transfer and flow field results are in good agreement with each other. The peak locations of local turbulent intensity and heat transfer regions overlap at the exact same location for both Re=500k and Re=662k cases respectively.

The overall results show that the heat transfer peak location on a gas turbine combustor liner strongly depends on the peak location of turbulent intensity of the swirling flow and not the maximum value of total velocity. In addition, the results also show that the peak locations are unaffected by the flow Reynolds number or flow rate. This is true within the tested range of Re=500k and 662k.
CHAPTER 1: INTRODUCTION

Gas turbine engines are an engineering development that has served mankind for over six decades. Both industrial sector and government research institutions have invested, and still are investing millions of dollars in improving gas turbine performance. As technology advanced throughout the years, designers have been able to push the combustion process further and closer to the stoichiometric limit with the help of high temperature materials and advanced analytical and fabrication tools. However, this positive progress has also created more challenges to engineers and scientists. There are still many issues requiring greater improvement and better solutions.

1.1 Cooling Systems

Maximizing energy output of gas turbine engines will result in increased heat loads on hot gas path components that will ultimately lead to premature failure of materials and higher NOx emissions. Several studies have claimed that today’s available technology and techniques are capable of pushing the thermal efficiency higher than ever and also doubling the engine power output with better control of air-fuel ratio (AFR) and flame temperatures. However as the thermal efficiency and power output increase, turbine inlet temperature will also increase consequently from 1700K to 2200K, which is higher than the blade material melting temperature at around 1850K. Over these years, many solutions have been proposed; using thermal barrier ceramic coating (TBC) and various other cooling techniques such as film cooling, back-side cooling, jet impingement cooling, etc. to remove as much heat load as possible from engine components and allow for longer life and better durability of components. However, excessive removal of heat from gas
turbines is unfavorable as it will reduce thermal efficiency because the heat generated is the needed energy. Therefore a proper design of cooling system is important to better compensate for both thermal efficiency and destructive heat loads.

For over three decades, film cooling has been used successfully in conventional combustors. The typical film cooling system used in conventional turbine combustor is as shown in Figure 1.1. With this technique, secondary coolant air is injected through holes or slots into mainstream air flow in primary combustion chamber to form a thin separate coolant air film or jacket so as to prevent the mainstream hot air stream from direct impingement on the combustor liner walls which otherwise causes enormous heat loads. Film cooling technique effectively lowers the flame temperature and increases the liner pressure drop to promote better mixing. Hence, this approach not only helps to improve fuel atomization, but also to eliminate localized hot spots and to reduce residence time.

![Figure 1.1: Typical Combustor Liner Wall Film Cooling](image)
However, the downside of this cooling technique is that the secondary coolant air stream will interfere with the combustion AFR by mixing itself into the mainstream air flow and unfavorably upsetting the AFR. Therefore it will create a non-uniform pattern factor causing non-uniform temperature distribution, induce unwanted thermal stress, and also cause premature failure of materials. Furthermore, the amount of liner wall cooling air added into the primary zone will significantly provoke the Carbon Monoxide (CO) emissions. At the near wall region where the wall cooling air is employed, the temperature of air will get relatively low such that all chemical reactions cannot be possible. However, the temperature of air at the center core of the primary zone will get so high that it results in higher NOx emissions. These localized hot spots can generate 25-30 ppm NOx and localized cold spots can also result in higher CO in the exhaust. Thus, even when the overall combustion zone is well maintained below the NOx formation threshold temperature range (1500-1650°C), the NOx levels will still be affected due to these localized hot and cold spots. This situation will therefore produce the worst overall combined emissions levels of both CO and NOx, which is undesirable. Thus engineers have paid a lot of attention at improving the uniformity of the air-fuel mixture so as to improve the temperature uniformity within the combustion primary zone and to minimize the formation of localized hot and cold spots.

1.2 Pollutant Emissions

In the past decade, the world has witnessed an incredibly rapid growth in the usage of gas turbine engines, replacing almost all other energy producing large machines. Since air route has become a more convenient and efficient mode of travel and transport in almost
every aspect of our daily life, the aircraft fuel consumption has literally become one of the biggest concerns in recent years. Likewise, stationary land-based gas turbines have also become the prime movers in the gas and oil industries. The outcome is so overwhelming that it has started to become a major problem to the world with its negative effect - pollutant emissions.

The pollutant emissions control is also a great challenge to gas turbine engineers in addition to designing effective cooling systems. During the past three decades, pollutant emissions from combustion processes have aroused public attention due to their adverse impact on health and environment. The environmental regulations governing pollutant emissions have become increasingly stringent with the requirements of controlling gas turbine emissions to produce lower levels of NOx, carbon monoxide, and unburned hydrocarbons.

The present Environmental Protection Agency (EPA) regulations have promulgated increasingly more restrictive emissions standards for NOx limits. Many studies have shown that the NOx emission level depends exponentially on flame temperature for both gaseous and liquid fuels. Other factors such as residence time in the primary zone, inlet air temperature, combustion pressure, equivalence ratio, pattern factors, and fuel atomization, might also have significant influences on NOx emissions production level. However, the influence of primary-zone flame temperature is still regarded as the more direct and substantial element to NOx emissions issues.

Therefore in an attempt to reduce NOx emissions, it is important to lower the flame reaction temperature and to eliminate localized hot spots from the combustion reaction zone. Consequently, the residence time for the formation of NOx should also be
minimized. Unfortunately, lowering the flame temperature will promote the formation of CO and UHC. In general, the formation of Carbon Monoxide (CO) and unburned Hydrocarbons (UHC) are highest at low-power conditions and decreases with an increase in power as opposed to NOx and smoke, which are fairly insignificant at low power conditions and appear to be the highest at high-power conditions.

Figure 1.2: Influence of Primary-Zone temperature on CO and NOx emissions [2]
With conventional combustors, the primary zone flame temperature can range from 1000K at low power condition to 2500K at high power operation. Figure 1.2 shows that when combustion temperature goes below 1670K, the CO and UHC are formed significantly due to incomplete combustion from the lack of sufficient oxygen and also due to the dissociation of \( CO_2 \); whereas at temperature higher than 1900K, excessive amounts of NOx are produced. This means only within a narrow band of temperature between 1670K and 1900K where the CO and NOx emissions levels are within the acceptable values. Based on the combustion efficiency point of view, the best compensation point for minimum productions of CO and NOx emissions is at the primary zone equivalence ratio of 0.8.

Another main objective besides controlling flame temperature to lower pollutant emissions is the uniformity of local temperature distribution. Even with the overall averaged flame temperature properly controlled within the desirable temperature range between 1670K and 1900K, any localized hot and cold spots are hardly predictable and may cause the local temperatures in separate regions within combustion zone to rise higher than 1900K producing NOx emissions and to drop lower than 1670K forming CO and UHC respectively. Thus, it defeats the purpose of narrow band flame temperature control and seriously affects the pollutant emissions levels making the flame temperature control effort become ineffective.

One effective way of better controlling both the flame temperature and AFR, is to use less liner wall-cooling air, especially in the primary zone. This will not only reduce local non-uniformity but also provide more accurate prediction and better control to mainstream flow temperature.
1.3 Water/Steam Injection

The addition of secondary air into main air stream is effective for lowering flame temperature but it can only be used sparingly because it raises the primary zone velocity, which will affect the ignition and stability performance negatively. For land-based stationary engines, large amount of water or steam are available; injection of which into the combustion zone is an alternative approach of formation of heat sink. However, this technique is clearly inappropriate for aircraft engines due to the non-availability of water or steam. Both water and steam injection approach are very effective in reducing NOx emissions and have been used on stationary engines since the early 1970s but unfortunately they do have some drawbacks [2].

With water and steam injection techniques, additional fuel is needed to heat the water up to boiling or superheated temperature, which causes an increase in fuel consumption by 2-3%. The water also needs to be extremely pure to prevent deposits and corrosion of components within the combustor. Therefore an extra cost is added to the capital due to the treatment of getting pure water, which might be costly to invest in water treatment facilities. Furthermore, water or steam injection will also result in higher CO and UHC emissions due to localized cold spots causing incomplete combustion, and increase in combustion pressure pulsations. With these drawbacks of water and steam injection, the “dry” solution has clearly become the better replacement for it [2].

1.4 Dry Low Emission (DLE) Combustor

The new generation of modern combustor is known as “Dry Low Emission” combustor (DLE) which has been proven to produce the emission level of NOx, CO, and
UHC as low as 9ppm. The DLE combustor operates utilizing premix gaseous fuel lean-burn combustion active control technique, which stages the injection of the fuel-air mixture to carefully control the location and the sequence of delivery within the combustion chamber. Additionally, it implements complex array of proprietary fuel-air-nozzles and combustion system geometry to effectively maintain the combustion flame temperature at near-optimum AFR within a narrow band of operating flame temperatures (1500~1650°C) which gives the lowest overall production level of NOx, CO, and UHC emissions. Nowadays, most of the DLE combustors offered by the OEMs, such as GE, Siemens, Alstom and Solar Turbines, guarantee low NOx emission level within the 9-20 ppm range. Figure 1.3 shows an example of modern dry low NOx emission combustor used in Solar MARS engines designed by Solar Turbines, Inc., which is recognized as one of the pioneered designers in modern DLE combustor.

DLE combustor not only uses the “Dry” solution which requires no water or steam injection into the main chamber, it also does not compromise in performance as well as reliability.

Lefebvre [2] has indicated that all gas turbine combustors must satisfy the basic requirements listed as below:

1. High combustion efficiency
2. Reliable and smooth ignition
3. Wide range of flame stability limits
4. Lower pressure loss
5. An outlet temperature distribution (pattern factor) that is tailored to maximize the life of turbine blades and NGV
6. Low emission

7. Freedom from pressure pulsations and instabilities

8. Geometry

9. Minimum cost and manufacturing issues

10. Maintainability and Durability

11. Multifuel capability

Figure 1.3: Solar Turbines MARS Low-NOx Combustor
Based on the requirements and challenges indicated above, in order to compensate for both pollutant emissions regulation standards and need for better efficiency, the modern dry low emission (DLE) combustors are believed to be the best solution for the next generation.

1.5 Focused Cooling

As previously stated, modern DLE combustor liners are designed to operate at minimum usage of film cooling if not completely without it. Designers have considered back-side impingement cooling as the solution for modern DLE combustors. Full coverage cooling across the combustor length is just not feasible for the application as the amount of cooling air is limited. However, if accurate heat load distribution on the liners can be obtained, then an intelligent cooling system can be designed to focus more on the localized hot spot(s) and provide more cooling to the deserving local hot spot(s) and reserve unnecessary coolant air from local cold spot(s). Thus, a highly efficient cooling system with focused cooling configurations is essential; which will certainly play an exceptional critical role in improving gas turbine engine efficiency and meeting the emissions standard regulations.

1.6 Swirler

The airflow pattern in the primary-zone plays a very important role in flame stability. The creation of toroidal flow reversal that entrains and recirculates a portion of the hot combustion products is commonly employed to enhance the mixing quality of incoming air and fuel. These highly dissipative vortices are continually refreshed by air
admitted through holes pierced in the liner walls, supplemented in most cases by air flowing through swirlers and flare-cooling slots, and by air employed in atomization [2].

One of the most effective ways of inducing flow recirculation in the primary zone is to fit a swirler in the dome around the fuel injector. Vortex breakdown is a well-known phenomenon in swirling flow. It causes recirculation in the core region when the amount of rotation imparted to the flow is high. This type of recirculation provides better mixing than is normally obtained by other means such as bluff bodies. It is because the swirl components produce strong shear regions, high turbulence and rapid mixing rates. These characteristics of swirling flow have been recognized and been used in many practical combustion devices to control the stability and intensity of combustion and the size and shape of the flame region [2].

There are two main types of swirlers widely used in combustors: axial and radial swirlers. Each kind of swirler can be fitted with flat vanes or curved vanes. Experience has shown that the flow fields generated by both the axial and the radial swirlers are broadly the same. Thus, the design rules established for axial swirlers can provide useful guidance in the design of radial swirlers. Beer and Chigier have proposed to use Swirl Number, a non-dimensional parameter to characterize the amount of rotation imparted to the axial flow [2].

\[
S_N = \frac{2G_m}{(D_{sw}G_r)}
\]

where \( G_m \) = axial flux of angular momentum

\( G_r \) = axial thrust

\( D_{sw} \) = outer swirler diameter

If swirl number is less than around 0.4, no flow recirculation is obtained and the swirl is therefore described as weak. On the other hand, for swirl number greater than 0.6,
the swirl is described as strong and most swirlers are practically operated under such conditions.

One of the primary functions of swirler is to generate a strong recirculation zone so as to induce combustion products to flow upstream to meet and merge with the incoming fuel and air. Therefore the size of recirculation zone has the direct positive impact on mixing quality of fuel and air. Kilik has done the most comprehensive investigation on the size of recirculation zone with different geometrical effects and has concluded the results as the following [2]:

The size of recirculation zone is increased by:

1. an increase in vane angle;
2. an increase in the number of vanes;
3. a decrease in vane aspect ratio;
4. changing from flat to curved vanes

In addition, Kilik showed that curved-vane swirlers induce larger reverse mass flows than the corresponding flat-vane swirlers and that the reverse flow is increased by an increase in the swirl number. Many evidences have proved that curved vanes are aerodynamically more efficient than flat vanes. This is because the curved vanes allow the incoming axial flow to gradually turn, which inhibits flow separation on the suction side of the vane. Thus, more complete turning and higher swirl- and radial-component velocities are generated at the swirler exit, which results in a larger recirculation zone and a higher reverse flow rate [2].
1.7 Literature Survey

There are several published articles that focus on the development of DLE combustors for industrial gas turbine engines. Studies [3-10] have focused on development of low NOx combustors with the viewpoint of producing lower emissions. These combustors are different in designs as some are annular, can, or silo type. However, one common feature for all these combustors is almost minimal usage of film cooling. There are other studies that focus on the liner cooling with different cooling configurations. Ferrera et al. [11] studied both convective and film cooled combustor analysis. They presented an analysis tool that analyzed the heat transfer on the backside of the liner. Smith and Fahme [12] focused on liner designs without film cooling for low emission combustors. Their design was mostly rib turbulated channel type flow cooling. Ling et al. [13] studied the cooling of the transition section behind the combustor using film cooling for a DLE combustor. Arellano et al. [14] present a study on an effective backside cooling scheme for an ultra-lean premixed combustion system. The Augmented Backside Cooled (ABC) liner eliminates film cooling in the combustion primary zone and uses trip strip turbulators along the cold side of the liner to enhance heat transfer and also thermal barrier coatings on the gas side liner wall to reduce heat load. The exit cone is film cooled and also contains dilution holes to produce the required combustor outlet temperature profile. They also used a variable geometry (VG) fuel injector to effectively control the amount of air entering the primary combustion zone area. Figure 4 shows the VG fuel injector used in their study. Bailey et al. [15] presented an annular combustor liner cooled by impingement jet rows with rib turbulated channel flow downstream of impingement. This is a typical cooling scheme for DLE combustor with a combination of impinging jets and rib turbulated
enhanced convective cooling. However, there are no published studies on measurements of heat transfer distributions on the gas side liner surface. This will be the first proposed study to focus on the gas side heat transfer under realistic combustor flow conditions.

There are several other studies that focus on the effects of combustor generated flow and temperature conditions and their effect on downstream NGV and rotor blades. These studies are mostly based on liner that have full coverage film cooling and pattern factors typical of aircraft engine or aero-derivative engine combustors. [16-17].

1.8 Experimental Objectives

As discussed earlier in the previous sections, it is clear that the “Dry Low Emission” combustors are a better solution for compromise between emissions level and performance as compared to conventional combustors. With minimum film cooling introduced, back-side wall cooling should be able to cool the combustor wall acceptably and adequately. Moreover due to the availability coolant air is limited, the focused cooling approach is needed to properly distribute the cooling supply based on the “regional” demand within a combustor. It is therefore extremely important to find out the actual heat load distribution on combustor liner walls and the exact location of highest temperature peak(s) at different flow conditions with the swirler provided by Solar Turbines Inc.

It is also very interesting to see how the flow characteristics, such as mean velocity, different velocity components, flow turbulence intensity, and turbulent integral length- and time-scale, can be related to the heat transfer experimental results. With an inadequate resource available from a limited number of publications to date, it certainly is a challenging problem.
The objectives of this project are to study the heat transfer distribution on the liner wall and flow characteristics, so as to determine the maximum heat load location and compare it with turbulence intensity peak location. The results obtained at different Reynolds number would be analyzed to study the effects of high-angle swirler and understand the fundamental physics of these phenomena.

Since most experiments are conducted at conditions suitable for accurate testing and not actual engine conditions, dimensionless parameters must be used to scale the results. The following equations show the definition of those important dimensionless parameters that were used in this investigation.

Reynolds number: \( \text{Re}_D = \frac{\rho U^2 / D(\text{inertia})}{\mu U / D^2(\text{viscous})} = \frac{\rho UD}{\mu} \)

Reynolds number is defined as the ratio of inertial and viscous effects and is influenced by fluid properties (viscosity and density), flow conditions (velocity) and geometry (relevant length scale).

Nusselt number: \( \text{Nu}_D = \frac{q''(\text{convection})}{q''(\text{conduction})} = \frac{hD}{k} \)

Nusselt number is defined as the ratio of convection heat transfer to fluid conduction heat transfer under the same conditions.
CHAPTER 2: EXPERIMENTAL APPARATUS

The complete experimental setup consists of inlet chamber with scaled up swirler, main combustion chamber with measurement equipment accessibility, exit nozzle, ventilation duct system, high capacity variable speed fan, and exit dump diffuser. The overview of experimental setup is shown in Figure 2.1.

![Figure 2.1: Overview of Experimental Setup](image)

2.1 Air Supply

A high capacity variable speed AC motor blower was used to supply sufficient high volume capacity air flow required for the experiment at desired Reynolds numbers. AC TECH variable speed motor controller provides fine and precise adjustment of fan speed frequency with a dial pad control panel. Figure 2.2 shows the setup of the blower mounted
securely on top of a high lifted rack for better security and safety purposes. Based on the factory recommended safety limit, the maximum rotational frequency of 53Hz was the limit, which equals to the maximum rotational speed of 2141rpm. The relationship between volumetric flow rate (CFM), rotational frequency (Hz), and pressure drop across blower (inH2O) was obtained through a series of tests as plotted in Figure 2.3.

During the experiments, measurements were recorded at two Reynolds numbers (500,000 and 662,000), which corresponds to 30Hz and 40Hz fan speed frequency respectively. From the calibrated fan performance curves, the volumetric flow rates of 8300CFM and 11000CFM were obtained with a pressure drop of 3 inH₂O and 5.15 inH₂O respectively across the blower. The corresponding velocities and Mach numbers can be back-calculated from volumetric flow rates and required information of basic properties of air as well as combustor geometry. The hydraulic diameter of the swirler, air velocities and Mach numbers for both the cases were calculated as the following:

Outer and inlet diameters of swirler: \( D_o = 16\text{in} \quad D_i = 8.75\text{in} \)

Effective cross-sectional area: \( A = \frac{\pi}{4} (D_o^2 - D_i^2) = 0.091m^2 \)

Effective perimeter: \( p = \pi(D_o + D_i) \)

Hydraulic diameter: \( D_h = \frac{4A}{p} = 7.25\text{in} = 0.184m \)

Main combustor chamber cross-sectional area: \( D = 40\text{in} \quad A_m = \frac{\pi}{4} D^2 = 0.811m^2 \)

Velocities at 8300cfm and 11000cfm respectively:
\[ v = \frac{Q}{A} \quad v_1 = 43.083 \text{m/s} \quad v_2 = 57.097 \text{m/s} \]

Since \( \gamma = 1.4 \quad R_g = 286.9 \frac{J}{kg\cdot K} \quad T = (21.4 + 273)K = 294.4K \)

Therefore the local speed of sound: \( a = \sqrt{R_g T} = 343.87 \text{m/s} \)

Local Mach number of air at 8300cfm and 11000cfm respectively:

\[ M = \frac{v}{a} \quad M_1 = 0.125 \quad M_2 = 0.166 \]

It was required that the Mach number in the combustor liner be in the range of 0.1 to 0.2 to ensure proper engine matching for both flow Reynolds number and Mach number. The scale of 5 from engine conditions provides similar flow close to engine operation to ensure the validity of the results.

Figure 2.2: AC TECH Variable Speed AC Motor High Capacity Fan
2.2 Inlet Chamber and Swirler

The swirler was a scaled up model of the swirler from Solar Turbines MARS engine. The 20-vane axial flow swirler was used to impart high degree rotation of flow at combustor primary zone which helps to promote better air-fuel mixing and to induce a recirculatory flow in the primary zone. The scaling, as stated before, was done to match both flow Reynolds number and Mach number through swirler. The swirler was retrofitted at the entrance of the combustor. A fine wire-mesh screen was installed at the very-front end of the inlet chamber to act as a filtration device and to help settling intake air flow and make it uniform. Figure 2.4 and 2.5 show the swirler setup mounted concentrically within the inlet chamber that connects to main combustion chamber.
Figure 2.4: Combustion and Inlet Chambers with Swirler Provided by Solar Turbines, Inc.

Figure 2.6 shows the 3-D solid model of scaled up swirler that was created using Solidworks CAD software. Basic information regarding the swirler geometry such as inlet vane angle, outlet vane angle, and diameter of turbulent flame stabilizer rods can be provided to give a good picture of the model swirler that was tested. The inlet vane angle was oriented at zero degree from axial plane and the outlet vane angle was designed to have 77.36 degree from axial direction. Figure 2.7 shows the cutout of swirler and basic dimension information in diameters. During the real lean-burn engine operation, gaseous fuel is injected from a series of fuel nozzles mounted on bluff bodies to premix with main stream intake air. The premixed gaseous fuel-air mixture is then ignited and the flame is stabilized at the recirculation zone behind bluff bodies.
Figure 2.5: Solar Turbines Swirler

Figure 2.6: 3-D Solid Model of Solar Turbines Swirler
Due to the wakes or recirculation vortices, which are caused by turbulent flow boundary layer separation on the surface of bluff body, the flow will be transitioned to highly turbulent so as to provide more energy to the flow and also to help in better air-fuel mixing quality. Moreover, the recirculation zone behind bluff body will contribute as a flame stabilizer to help trapping the flame at high speed flow condition. Therefore, the swirler model tested in this project was equipped with twenty cylindrical rod bluff bodies, or spokes, with a diameter of 5/8-in as flame stabilizers and vortex generators.

The expansion ratio of flow from swirler to main combustion chamber can be determined in various ways depending on definition, such as using the hydraulic diameter of swirler calculated in the previous section, i.e. 7.25inches, or with the outer diameter of the swirler, i.e. 16 inches.

Flow Expansion ratio: \( r = \frac{D_h}{D} = \frac{7.25}{40} = 0.181 \) 
\[ r = \frac{D_o}{D} = \frac{16}{40} = 0.4 \]
2.3 Main Combustion Chamber

As pointed out earlier, the combustor test model was designed to be scaled up 5 times of the original size combustor; from 8”-diameter to 40”-diameter with a length of 8.5-foot so as to provide higher measurement resolution and better accessibility.

![Figure 2.8: IR Camera Windows Cutoff Locations](image)

The entire section was fabricated using 16-gage galvanized sheet metal. The sheets were cut to length and rolled to desired diameter and spot-welded onto the pre-made steel flanges to retain the shape. Windows for the IR camera were cut with equal spacing (14.57 inches) at six different locations along the main combustion chamber for heat transfer experiments as shown in Figure 2.8. The camera mounting brackets were welded on to firmly secure the IR camera that weighs about 5lbs. On the opposite side of windows, a stainless steel foil surface heater assembly was mounted with a well-insulated wooden block from the back, the space in-between them filled with thermal-insulation material to minimize heat loss. During the experiments, when camera was mounted in one of the windows, the others were sealed off by strong magnetic sheets to prevent air leakage.
For the flow experiments, hot-wire constant temperature anemometry (CTA) system was used to measure flow characteristics. It is important to note that the hot-wire probe requires a solid probe support to avoid vibration or fluctuation so as to prevent undesirable noise and incorrect measurements. However, the original factory probe support from TSI was not suitable for this application. Therefore, a much stronger custom-made probe support was fabricated to provide the best integrity and geometrical advantages such as an adjustable length that has the ability to work as telescope.

The probe assembly was mounted on top of combustor which hung down vertically into the chamber. A long straight slot was cut axially across the combustor as positioning rail with sixteen pre-marked locations. Rail supports were welded-on along side of the rail with lock-pin holes to help locking down the probe support with minimal play. Each pre-marked location and hole is 6” apart from each other as shown in Figure 2.9.

Figure 2.9: Hotwire Probe Mounting Rail on Top of Combustor with Pre-Marked Locations
2.4 Exit Dump Diffuser

A simple exit diffuser was built at the end of the system to dump the discharged air and free the backpressure to provide less restriction and better air flow as in Figure 2.10. Since air is discharged within the building near walkway, the exit diffuser was purposely designed to slow down the discharged air and to avoid noise pollution.

Figure 2.10: Exit Dump Diffuser

2.5 Heat Transfer Experiment

2.5.1 FLIR SC500 Infrared Thermal Imaging System

The FLIR SC500 Infrared Camera was used to capture the liner wall surface temperature distribution within combustor as shown in Figure 2.11. The SC500 camera is a focal plane array system type IR camera using microbolometer as detector material and
has thermal sensitivity as high as within 0.1ºC at 30ºC. The camera has a maximum resolution of 320x240 and wide measurement range of -20 to 500 degree Celsius with an accuracy of only 2% or 2 degree Celsius. With a proper filter and calibration, an even larger measurement range of -20 to 1500 degree Celsius can be obtained. The target surface emissivity can be precisely calibrated with its full emissivity adjustment from 0.1 to 1.00. The refresh frequency of imaging can be set to as fast as up to 60Hz. Multiple color and B&W display palettes are available for user’s convenience and applications.

Figure 2.11: FLIR SC500 Infrared Camera (left); IR Imaging Temperature Acquisition System (right)

Simple surface temperature calibration can be conveniently carried out by using the surface emissivity adjustment feature, which is one of the unique features in the program. With high-precision thermocouples measuring the surface temperatures within a selected area, line, or even point, the IR temperature values can easily be calibrated by correcting the surface emissivity until the IR readings has equaled to the known desired temperature.
values obtained from thermocouples. Depending on the surface properties but in general, if the surface temperature is underrated then the surface emissivity should be lowered, and vise versa for overrated temperature.

2.5.2 INSTRUNET Temperature Measurement System

InstruNET RTD temperature measurement system consists of a data acquisition hardware, “i100-box”, and application software, “InstruNET World software”. The i100-box provides 8 differential analog input voltage channels and 8 voltage output channels that can be used for RTD measurement with the help of one user-supplied external shunt resistor. The A/D resolution processes data signal at 14-bit with the A/D conversion time as low as 4 microseconds. For our application, a series of high-sensitivity fine gage K-type thermocouples were used to measure the upstream and downstream temperature of air within the combustor, the air before entering the swirler and the air before exiting the exit nozzle respectively.

Using the InstruNET software, an analog temperature signal can be monitored and managed in a real-time view window, and can be recorded and saved in the computer disk in spreadsheet format for post-analysis purposes. According to the reference of temperature accuracy on the InstruNET website and actual error measurement in the laboratory, the temperature accuracy fluctuation was found to be around +/- 0.5ºC for the K-type thermocouple operated at 15-100ºC range [30].

The system can be conveniently calibrated using temperature simulator. The offset and gain-scale corrections can be manually adjusted to compensate for errors possibly induced by difference in resistance due to the length of thermocouple wires.
2.5.3 Wall Surface Heaters

The wall surface heaters were specifically made to suit the needs of experimentation requirements. Figure 2.13 shows the basic schematic of wall heater system construction and energy processes for steady-state experiments. The inner face of heater inside the combustor is coated with thin flat black paint. The other face is glued onto a backside supporting wooden block by using a thin special double-sided clear adhesive. The outer face of the wooden block is properly insulated with building insulation materials. Figure 2.14 shows the multilayer construction of the wall surface heater system.

Based on our calculations, in order to obtain a reasonable difference in temperature to ensure lower uncertainty, not to overheat the heaters, and to reduce waste of available electrical power, the maximum temperature difference of 20°C was determined as the best break-even point and therefore designated as one of the design criteria for the heater system. This value was then used to calculate the required resistance for our application and limitation of available electrical power in the laboratory.
Figure 2.13: Wall Surface Heater System

Figure 2.14: Construction Layers of Wall Surface Heater System
The required resistance for heater to produce $200\, W/m^2$ of maximum heat flux was calculated to be 29.3 ohm. For this application, the 0.0005" Type 304 stainless steel shim stock, as shown in Figure 2.15, was used as heater foil material, which has a thermal conductivity of $16.2\, W/mK$ and electrical resistivity of $7.2\times10^{-7}\, \Omega m$.

![Figure 2.15: Stainless Steel Shim Stock for Heater](image)

Using the Ohm’s law, with the known resistivity, resistance as well as the thickness of foil, the vertical length and width of heater foil strips were determined to be 12 inches and 1 inch respectively with the horizontal length of 30 inches in axial direction, for best
compromise between voltage and current, which resulted in 360 inches overall length. The relationship can be found expressed as the following:

\[ R = \rho \frac{L}{A_{ss}} \quad A_{ss} = wt \]

Due to the large coverage area, it is technically difficult to hand-cut the stainless steel shim stock. In order to reduce the probability of making mistake and having to re-cut everything, the system was divided into three separate sections identically. Each section was made individually and equally with same resistance. All three sections were then connected (soldered) together with copper strips in series. These heater foils were then installed firmly onto the liner wall surface and painted with flat black color to simulate black body surface condition for IR imaging application. The special paint for the application is rated as high temperature heat resistance with flat dull finish, which is designed for barbeque grill pit as shown in Figure 2.16. The flat non-shiny finish is crucial for the application in order to prevent reflection of wave causing confusion to IR camera resulting in temperature measurement error.

The heater foils were also properly insulated to minimize heat loss through the back of the system. However, a small amount of heat loss was unavoidable and was then measured at different temperatures so as to account during data processing calculation to provide more accurate results. Figure 2.17 shows the relation of conductive heat loss through wall that is caused by the temperature difference between heater and wall.

The heat generation from heater can be adjusted through the use of transformers, as shown in Figure 2.18, by varying the voltage and amperage to obtain the required electrical power output.

\[ \bar{Q} = P_E - \bar{Q}_{loss} \quad P_E = IV \quad V = IR \]
Figure 2.16: Hi-temp Heat Resistant Flat Black Paint

Figure 2.17: Conductive Heat Loss through Wall due to Temperature Difference
Voltages and Current Amperages were measured using high precision multimeter and clamp-on amp meter. The accuracy of voltage and amperage measurements is within ±0.001V and ±0.001A respectively. Since the experiments were performed using steady-state method, therefore the real-time voltage and amperage readings were not required once the values were decided and set. However, it was constantly checked after every run to ensure the consistency of results.

2.6 Flow Characteristic Experiment

2.6.1 TSI IFA300 Hot-wire Constant Temperature Anemometry

TSI IFA 300 Hot-wire Constant Temperature Anemometry (CTA) is a point-measuring technique used to measure the velocity at a point and provide continuous
velocity-time series, which can be processed into amplitude and time-domain statistics. This technique is particularly suitable for measurement of flows with very fast fluctuations at a point (high turbulence) and the study of flow micro structures [28].

The working principle is based on the cooling effect of flow passing over a heated wire. The heat transfer mechanism of hot-wire anemometry can be explained as the convective heat transfer from a cylinder. The basic relation between \( Q \) and \( U \) for a wire placed normal to the flow can be expressed as:

\[
Q = (T_w - T_0)A_w h = A + BU^n;
\]

where \( n \approx 0.5 \)

- \( A_w \) = wire surface area
- \( h \) = heat transfer coefficient
- \( A \) and \( B \) = calibration constants

The convective heat transfer \( Q \) from a wire is a function of velocity (U), temperature \( (T_w - T_0) \), and the physical properties of the fluid \( (\kappa, \rho, \mu, \alpha) \).

The system principle employs a Wheatstone bridge circuit with one arm connected to probe sensor wire with resistance \( (R_w) \) that is heated by electrical current as shown in Figure xx. The Wheatstone bridge is used to measure an unknown electrical resistance by balancing two legs of a bridge circuit, one leg of which includes the unknown component. With the use of Galvanometer for detecting zero current to extremely high accuracy, the Wheatstone bridge is well suited for the measurement of small changes of a resistance due to its sensitivity. Therefore, the hot-wire CTA is claimed to be one of the best techniques for high precision velocity fluctuation measurements [28]. Figure 2.19 gives the overview of hotwire CTA system and its circuit working principle.
A cautiously and properly conducted calibration of the sensors is extremely important during the calibration processes in order for the system to work properly with the sensors. The TSI factory calibration step-by-step procedures must be followed in order to obtain the right calibration curve. The TSI hotwire air calibrator was used to perform the calibration process more easily and appropriately. The angle rotation arms and bracket equipped on the calibrator provides the designated angle increments and height adjustments for different types of sensors depending on user’s needs and applications.
For this project, the TSI model 1128B air velocity calibrator was used to perform the calibration of the sensors used in the experiments as shown in Figure 2.20. The 1128B model calibrator is manually operated, bench-top system designed to make it suitable for variety of single-sensor, dual sensor, and even triple-sensor. The calibrator consists of a settling chamber, secondary in-line nozzle, air pressure regulator with dry/wet filter, pair of coarse and fine adjustment valves, adjustable probe manipulator, thermocouple, and differential pressure transducer.

The settling chamber is equipped with a free jet nozzle with choices of diameter of 10mm or 14mm. Several choices of secondary in-line nozzles can be installed and combined with the free jet nozzle for increased sensitivity and accuracy at low velocity. In these experiments, the velocity range was estimated from 1 to 40 m/s, therefore the nozzle combination of 10mm exit free jet nozzle and #1 secondary in-line nozzle was used. The differential pressure transducer used in the system is a MKS Baratron Type transducer that has an output of 0 to 10 volts DC for differential pressure of 0 to 100mmHg. The A/D system has an input of -5 to +5 volts, so a signal conditioner was built into the IFA300 system to provide a 5-volt offset and gain (1 or 10) so as to give a better resolution at low pressure readings. The thermocouple is a copper-constantan Type-T unit and is mounted in the settling chamber upstream section of flow-straighten screens located inside the settling chamber. Figure 2.21 shows the overview of complete TSI IFA 300 CTA calibration system illustrating how the air calibrator system is connected to the IFA 300 CTA system.

A dual sensor X-probe was used for the experiments. The probe was calibrated to the velocity range of 0 to 50m/s. The calibration table was created with 17 calibration velocity data points. Since X-sensor probe can measure up to two velocity components,
therefore the yaw angle calibration is required if two velocity components measurements are desired. The IFA 300 calibration program allows user to tag up to eight velocity points for yaw calibration but most of the time only one value is enough for a reasonably accurate calibration. TSI actually suggests tagging the velocity value of about 40% of the full scale velocity range. Therefore, the velocity point of 20m/s was tagged for yaw velocity calibration. Eleven slant angles (maximum) with an increment of 6° were selected for yaw angle calibration.

![Diagram of TSI IFA300 CTA Calibration System](image)

Figure 2.21: Overview of TSI IFA300 CTA Calibration System [29]
It is important to note that the pressure transducer should be turned on and allowed to warm up and stabilize before use to prevent changes in values which occurs due to increase in temperature of electronic components before reaching steady-state. Figure 2.22 is the calibration curve obtained for the X-sensor probe used in the experiments.

Figure 2.22: Hotwire Calibration Curve
CHAPTER 3: EXPERIMENTAL METHODOLOGY

Since the experimental criteria were based on real engine conditions, several important dimensionless parameters were properly matched and characterized so that the experimental results can be applicable to validate the real engine operation investigation. Figure 3.1 shows a brief general idea of flow behavior within the combustor.

![Figure 3.1: General Idea of Flow Behavior within Combustor](image)

In real engine conditions, the core combustion temperature can get as high as 1850K, which is around the melting temperature of materials of typical turbine components without thermal barrier coating. The operating pressure within combustor can also be as much as 50atm. Under these critical conditions, it is not very feasible to perform the experiment realistically and it might also be too dangerous to be done in the laboratory. Moreover, the 8”-diameter real engine combustion chamber provides limited space, where it is not practical for various experimental equipments to fit in and to take measurements. Therefore, a non-reactive cold combustion investigation was conducted as a more realistic
and feasible option for this project. The intake air temperature was at room temperature and the operating pressure was the local atmospheric pressure.

Typically under real engine conditions, the combustor flow Reynolds number is in the range of 300,000 to 1,000,000. For this project, two Reynolds numbers (500k and 662k) were chosen to simulate engine operation conditions. Using similarity analysis, the relationship between actual heat transfer coefficient and test model heat transfer coefficient can be calculated as the following.

**Real Engine Conditions:**
- Air Temp. = 1850K
- $k_{air} @ 1850K = 0.124W/m.K$
- Operating pressure = 50atm
- Combustor diameter = 8”

**Test Model Conditions:**
- Air Temp. = 293K
- $k_{air} @ 293K = 0.0263W/m.K$
- Operating pressure = 1atm
- Combustor diameter = 40”

**Similarity Analysis:**

\[
(Nu)_{actual} = (Nu)_{test} \quad \left( \frac{hD}{k} \right)_{actual} = \left( \frac{hD}{k} \right)_{test}
\]

\[
h_{actual} = \left( \frac{k_{actual}}{k_{test}} \right) \left( \frac{D_{test}}{D_{actual}} \right) h_{test} = 4.71 \times 5 \times h_{test} = 23.57h_{test}
\]

Based on the calculations, the actual heat transfer coefficient at real engine condition should be about 23.57 times higher than the experimental results obtained in the laboratory.
3.1 Steady-State Temperature and Heat Transfer Measurements

The experiment was accomplished by sending cold air flow (room temperature) into main combustion chamber through the swirler mounted concentrically within the inlet chamber. The entire long liner wall section across combustor chamber was heated uniformly with six identical steel foil surface heaters to simulate constant heat flux boundary condition. The heaters are connected in series across the length of the combustion chamber. The heat flux can be adjusted with three variable transformers to obtain the desired temperature difference between the wall and the bulk air. The inlet upstream and exit downstream air temperatures were measured with high sensitivity fine gage K-type thermocouples using the INSTRUNET temperature measuring system and were later used to perform heat transfer calculations.

To start the experiment, the IR imaging system and INSTRUNET temperature system were initiated and re-calibrated to ensure accurate measurements. The blower air supply system is switched on and set to the desired air flow rate based on the required Reynolds number. Once the air flow speed was properly set, surface heaters are set to the required heat flux power to obtain the desired temperature values.

Since the experiment is performed using steady-state method, the entire set up is left to be running for at least 25-45 minutes depending on the ambient condition until a steady-state wall surface temperature is reached. The IR image of temperature distribution is acquired at six different locations along the combustion chamber. The raw temperature data from all six different images are merged and combined to compute the local heat transfer coefficient distributions. The conductive heat loss through wall due to temperature difference between liner wall and outside combustor surroundings is also taken into
account to obtain more accurate results. The basic convective heat transfer equation was used as shown below.

\[
h = \frac{Q}{A(T_{\text{wall}} - T_{\text{air}})} \quad \text{where} \quad Q = P_E - Q_{\text{loss}}
\]

\[
h = \frac{P_E - Q_{\text{loss}}}{A(T_{\text{wall}} - T_{\text{air}})}
\]

In the experiment, the wall was maintained at 15°C over the room temperature of 20°C. The target surface emissivity was calibrated using the INSTRUNET temperature measuring system and was determined to be 0.92. The fastest picture frame rate of 60Hz on the IR camera was applied to improve measurement accuracy.

**3.2 Hot-wire CTA Turbulence and Velocity Flow Field Measurements**

The TSI IFA 300 Constant Temperature Anemometry system is as shown in Figure 3.2 was used to perform the flow characteristic measurements. The film-type dual sensor 45° X-probe as shown in Figure 3.3 was used to measure the U and V component velocities with high turbulent fluctuations. The data sampling capacity was set at the sampling rate of 2000Hz and sampling size of 2kpts/channel. The flow characteristics were measured directly using the IFA300 system and was post-analyzed by utilizing the application software provided to obtain important flow properties, such as turbulent intensity, skewness coefficient, flow angle, shear stress, etc. The flow characteristics of the entire combustion chamber were measured from the near-wall surface all the way to the center of combustor,
i.e. 10 axial locations along the combustor and 20 radial heights towards the center of combustor, which has a total of 200 acquisition points.

Figure 3.2: IFA 300 Constant Temperature Anemometry Unit

Figure 3.3: Film Type Dual Sensor X-Probe
It is commonly understood that turbulence intensity may reflect the intensity of heat transfer rate. The area with high velocity turbulence intensity will generally be the area with higher heat transfer rate. Therefore, it is of our interest to pay more attention to flow turbulence intensity within combustor. Flow turbulence intensity is defined as following and a typical turbulent flow velocity profile is shown as in Figure 3.4:

\[ TI = \sqrt{\frac{\overline{u'^2}}{\overline{u}}} \]

where \( \overline{u} \) is mean velocity

\( u' \) is velocity fluctuation

\( \overline{u'^2} \) is the Root Mean Square (RMS) value of velocity fluctuation

### 3.3 Integral Length Scale Analysis

The turbulence integral length scale is a measure of average spatial extent or coherence of the fluctuations. For a particular point in the flow, the magnitude of integral
length scale is a function of not just the quantity but also of the direction of separation depending on which directional correlation is used. It is also the length scale or size of the largest eddy or simply the width of the flow.

The autocorrelation function is a correlation of velocity signal with itself, displaced by a time period, T.

\[ R(T) = \frac{(u'(t) \cdot u'(t-T))}{(\overline{u'})^2} \]

The integral of autocorrelation is used to compute the integral length scale by integrating the autocorrelation curve from zero out to the first zero crossing in order to capture both the finest and the largest scales and to obtain the integral time scale [25-27].

\[ J = \int_{0}^{n} R(T) \, dt \]

The integral length scale is the product of mean velocity and time scale.

\[ L = U \cdot J \]

As it was pointed out previously, the integral length scale is determined from the autocorrelation function, but often the legitimacy of autocorrelation function itself may be questionable if the spatial domain is not sufficiently large. In general, the spatial domain in an experiment is limited by the restrictions on experimental equipment or measurement instrumentation.

The key to calculating the proper integral length scales is to use an iterative process for choosing the desired sampling rates and times for the experiment. These choices lead directly to considerations for data storage and processing times, i.e. the spatial domain. The TSI IFA300 Anemometry software is capable of sampling up to 300 kHz and has a maximum block size of 256k. The software generates the autocorrelation by processing a
block of data and averaging the blocks into a composite autocorrelation. The preciseness
of the autocorrelation increases as the number of points per block increases. However, the
processing time also increases significantly as the number of points per block increases.
The maximum number of points per block allowed by the software, 256k, was used for the
autocorrelation of this experiment. The integral of autocorrelation is used to compute the
macro- and micro-scales, both of which are related to the size of vortices present in the
flow. An adequately large sampling rate is required to capture the micro-length scales, and
a longer duration is needed to capture the macro-length scales. Thus, defining an
appropriate sample frequency and duration is required to properly calculate the length
scales. An iterative process is needed to determine a compromise between the rate,
duration, and disk space size. The compromise should not sacrifice result accuracy, but it
should keep in mind the length of times needed for disk sizes and data processing times
[25].

The Power Spectral Distribution (PSD) is a function that displays the distribution of
the signal frequencies present in the sample. An accurate plot of the PSD requires that the
appropriate frequencies in the signal be captured; in essence, rapid sampling rates will
capture the smallest frequencies and long sampling durations will capture the large
frequencies. The PSD is developed by using a Fast Fourier Transform (FFT) on a specified
number of data points, or block of data, and the results from each block of data are
averaged into one combined spectrum. The power spectrum increases in accuracy as the
number of data points per FFT increases. The software used to process the data, provided
with the TSI IFA300 Anemometer, had a maximum block size of 256k. The PSD was also
used to determine the correct choice of a low-pass filter. Two times the filter setting
satisfies the Nyquist criterion; however, Simon et al. and Roach suggest using five times the low-pass filter setting to have more confidence in capturing a complete signal [25]. The typical form of autocorrelation function is such that it decreases rapidly to its first zero-crossing, after which it may become negative and proceed to oscillate about zero.
CHAPTER 4: RESULTS

4.1 Flow Data

Two-dimensional U-component turbulence intensity distributions of the flow inside combustion chamber for both Reynolds numbers of 500k and 662k are shown in Figure 4.1. The results confirmed the existence of strong vortices around upstream corner region and intense expanding flow impinging radically onto liner wall. The flow then bounced back towards axial core center and merged into mainstream flow through the exit. It is important to note that the locations where the expanding flows hit the liner wall were exactly on the same location for both Re=500k and 662k. Although the higher Reynolds number case shows a strong push-back of flow at the downstream section, the upstream sections for both cases were similar. The results predicted that the peak turbulence intensity location would be at the same spot regardless of Reynolds number.

![U-component Turbulence Intensity Distribution for Re=500,000 (top) and Re=662,000 (bottom)](image)

Figure 4.1: U-component Turbulence Intensity Distribution for Re=500,000 (top) and Re=662,000 (bottom)
In order to further bolster this result, the 1-D distribution of turbulence intensity at the very near wall surface was plotted in Figure 4.2 to show the peak location more closely. The results were clear and as expected; the peak locations for both cases of Reynolds numbers were on the same location.

Figure 4.2: Turbulence Intensity Distribution at the very near wall surface locations for Re=500,000 (top left), Re=662,000 (top right), and combined (bottom)
Two-dimensional U-component velocity flow field for both cases are shown in the Figure 4.3. Both cases share the similarity in air flow behavior by showing the highest and the strongest air flow rushing through swirler annulus opening outlet. The high speed air flow is concentrated around the region after the swirler’s exit and it generates high turbulent vortex zone at the center core. Although, it is obvious that the U-component velocity for higher Reynolds number case (Re=662k) must be higher than lower Reynolds number case (Re=500k), there was a distinguishable difference between them. The U-component velocity distribution for Re=500k was somehow maintained throughout the combustor until the downstream region even after passing the upstream region which is in sharp contrast with the Re=662k case where the U-component velocity decreases immediately after the upstream region and decreases continuously until the downstream region. These results can be interpreted in terms of swirling momentum.

Figure 4.3: U-component Velocity Flow Field for Re=500,000 (top) and Re=662,000 (bottom)
Due to the strong airflow for Re=662k, the swirling momentum does not die out moving through the end but with the weaker airflow at Re=500k, the swirling momentum dies out after the impact impingement on combustor wall. In other words, the air flow of Re=500k case was able to maintain higher U-component velocity throughout the combustor due to the lack of momentum in swirling effect. The swirling flow for lower Reynolds number was not able to keep its momentum until downstream and caused the flow to change direction merging into mainstream axial flow.

Figure 4.4: Total Velocity Flow Field for Re=500,000 (top) and Re=662,000 (bottom)

Now with the consideration of both U- and V-component velocities, the total velocity flow field can be determined by \(\sqrt{U^2 + V^2} \). However, it is very important to note that the “total” velocity found here is not the real actual total velocity for the flow within the combustor. Since the air flow dynamics inside a combustor are complex and three-dimensional, therefore three velocity components must be needed in order to actually
determine the total velocity. With two-dimensional probe CTA system, only two components can be obtained, thus the swirling or tangential velocity component was not measured directly from this experiment. Nevertheless, the swirling effect can still be visualized and proved for its existence from the results.

As shown in Figure 4.4, one can easily notice that the total velocity flow fields for both cases look essentially similar except the fact that the total velocity center core with high flow region for \( \text{Re}=662k \) looks like stretched out version of \( \text{Re}=500k \). The center core region for \( \text{Re}=662k \) was elongated further downstream towards the end as compared to the one for \( \text{Re}=500k \). However from the turbulent intensity distribution map and heat transfer results, the peak location was not shifted and occurred at the same location for both cases.

All flow experimental results so far have pointed to the same direction as we had expected, however there is still one more important parameter of flow characteristics left to be discussed. In the previous session, it was evident that turbulence intensity distribution and velocity flow field are useful when trying to understand the flow behavior, but they do not really help to describe the flow microstructures and time-dependent information of turbulence itself such as the size of eddies and life time of turbulence structures. Therefore both the turbulence integral length- and time-scale were determined to provide better understanding of flow turbulence characteristics. Turbulence integral length scale quantifies the sizes of turbulence eddies and integral length scale is commonly referred as a quantitative characterization of the time needed for a signal to de-correlate from the function.

The results of both integral length- and time-scale for \( \text{Re}=500k \) are shown in Figure 4.5 and 4.6. Due to the fact that the integral length- and time-scale measurements were
performed at very costly time-consuming process and were purely based on our personal interests, therefore only Re=500k case was investigated to give a great picture of flow turbulence information. Yet, the results were as expected and once more confirm the consistency and agreement of the distribution of turbulence intensity and integral length scale. The upstream expanding and swirling regions evidently show stronger turbulence with larger sizes of eddies and longer characteristic de-correlation life time which are also the regions that have the highest heat transfer coefficient.

The span-wise averaged turbulence length scale distribution as shown in Figure 4.7 was also plotted to help visualize the changes in turbulence characteristic showing how the energy of turbulence eddies was dissipated so drastically across the combustor. As mentioned earlier, the air flow for Re=500k did not carry strong enough swirling momentum throughout the combustor causing the swirling flow to straighten out quickly.
than the one for $Re=662k$. In other words, the energy contained within turbulence eddies was inadequate to sustain the turbulence eddy itself and therefore cause them to die out quickly before reaching through the end of combustor.

Figure 4.6: Turbulent Integral Time scale for $Re=500,000$

Figure 4.7: Span-wise Averaged Turbulence Integral Length Scale Distribution
4.2 Heat Transfer Data

Wall surface temperature distributions for different Reynolds numbers along the combustor liner are shown in Figure 4.8. Flow enters the combustor through annulus swirler and expands with high degree angle of swirl impinging on the combustor wall, therefore causing much lower temperature at upstream region than at downstream.

![2-D Temperature Distribution Map](image)

Figure 4.8: 2-D Temperature Distribution Map for Re=500,000 (top) and Re=662,000 (bottom)

From basic convective heat transfer theory, the heat transfer rate should therefore be higher at upstream and lower at downstream. The ambient intake air temperatures were measured at 21.2°C and 24.5°C for Re=500k and Re=662k respectively. The output voltages and currents were metered at 43V & 1.25A, and 44V & 1.3A during the experiments for Re=500k and Re=662k respectively. The output heat flux of a heater can be determined by the product of output voltage and current and divided by surface area of single heater. Heat flux for Re=500k and Re=662k were found to be 173.95 and 185.11 W/m² respectively of single heater with surface area of 0.309 m².
As shown in Figure 4.8, the temperature difference across the combustor gives as much as 4°C and about 2°C in difference between the highest and lowest values for Re=500k and Re=600k respectively. With ambient air temperatures of 21.2°C and 24.5°C, the temperature differences are relatively measurable as much as 19% and 8%. It indicates that there is a distinguishable localized heat transfer peak region within combustor. In addition, the percentage differences in both cases also imply that the temperature distribution spreads out more uniformly for Re=662k than Re=500k. As it can also be seen in Figure 4.9, the heat transfer distribution for Re=662k is steadier than Re=500k. Therefore the overall heat transfer coefficient for Re=662k is higher.

![2-D Heat Transfer Distribution Map for Re=500,000 (top) and Re=662,000 (bottom)](image)

On the other hand, it is also important to note that the upper part of the Figure 4.8 and 4.9 has shown some significant wash away effects. It is due to the swirling flow effect that was generated by swirler which also contributed to the additional heat transfer effect in the rotational direction. As the swirling flow was introduced in the clockwise direction, the
flow went around within combustor from top to bottom across the heaters. Thus, the bulk air was heated after passing the very top part of heaters and the bulk air temperature continues to increase causing the heat transfer coefficient to drop consequently.

This wash-away phenomenon not only proves the existence of high swirling flow in action but also the effect of bulk air temperature reducing heat transfer coefficient. Due to the wash-away effect caused by the high angle strong swirling introduced in the combustor, only the very top part of these 2-D distributions are “clean” which is unaffected by the heated bulk air temperature. Therefore, only the first row of the 2-D distribution was plotted in Figure 4.10 as 1-D profile to properly show the unaffected heat transfer distribution. Both profiles for Re=500k and Re=662k behave in very similar way except the values for Re=662k are generally a fraction higher than for Re=500k. Both cases show an obvious bump representing a positive local hot spot pertaining to the highest heat transfer coefficient and concentrated heat load.

The averaged wash-away effect was also plotted in Figure 4.11 as it was an interesting finding from experiments. The significance of the influence of the swirling effect is evident. Within a one-foot-wide heater surface (0 to 180pts in 12”), the heat transfer coefficients drop from 19.8 to 18 $W/\text{m}^2$ and 23.3 to 21.3 $W/\text{m}^2$ for Re=500k and Re=662k respectively, which are about 2 $W/\text{m}^2$ or 10%. These results evidently show the high swirl angle flow was dominating within the combustor over the axial flow. Therefore, it can be said that the swirler has in fact effectively created the high angle swirling flow within the combustor; just as it was expected to perform.
Figure 4.10: 1-D Heat Transfer Distribution for $Re=500,000$ (top) and $Re=662,000$ (bottom)
Figure 4.12 shows the direct comparison of heat transfer coefficient distribution for both Re=500k and Re=662k. It clearly shows that the area under the curve for Re=662k is larger and therefore indicates a higher overall heat transfer coefficient. The peak value for Re=662k is about 3 $W/m^2$ higher than for Re=500k. As the flow passes the peak region, the heat transfer coefficient for Re=662k holds steadily through the entire combustor until the exit; however for Re=500k, the curve drops continuously after the peak. The explanation for this phenomenon is fairly straightforward, which has been covered similarly in the previous flow result session. The strong airflow for Re=662k contains higher swirling momentum moving through the end, which results in maintaining higher heat transfer; whereas the weaker airflow at Re=500k lacks swirling momentum causing swirl to decrease significantly after the impact impinging on combustor wall and therefore results in a continuous decrease in heat transfer throughout the end. In other words, the heat transfer wash-away effect was less significant for Re=500k case as compared to Re=662k case. This result has again shown how consistent and good the agreement between the heat transfer and flow experimental results is.

Since the experimental conditions and results were scaled to match the real engine conditions, based on the similarity analysis, the real engine heat transfer coefficient was calculated to be 23.57 times of the experimental results. Therefore, it is believed that the peak values for both Re=500k and Re=662k should be at 707 and 636 $W/m^2$ respectively. These experimental results were also scaled to match the real engine conditions; the local Nusselt number of real engine condition can be calculated by the scaling factors.
Figure 4.11: Swirling Effects on Averaged Heat Transfer Distribution for Re=500,000 (top) and Re=662,000 (bottom)
Figure 4.12: Comparison of Heat Transfer Coefficient Distribution for Re=500000 and 662000

Figure 4.13: Comparison of Local Nusselt Number Distribution with respect to Combustor diameter for Re=500000 and 662000
Local Nusselt number distributions for both Re=500k and Re=662k were plotted. Figure 4.13 shows the local Nusselt number distributions with reference to combustor diameter for both cases. Similar to the heat transfer coefficient distribution curves, the difference in the peak values of Nu is about 100 and in the downstream section, the difference is 200. The Nusselt number for Re=662k seems to reach a steady value at around 825; whereas for Re=500k, Nu decreases continuously through the end.

The local Nusselt number distributions with reference to x (axial distance) were also plotted in Figure 4.14 to show the linear proportionality of local Nusselt number distribution with respect to axial distance. The steeper gradient of curve for Re=662k usefully indicates how the Nusselt number response favorably with increasing distance and also provides a better vision to distinguish the overall performance between both cases.

![Diagram of Local Nusselt Number Distribution](image)

Figure 4.14: Comparison of Local Nusselt Number Distribution with respect to axial distance for Re=500000 and 662000

62
The current design criteria of cooling system used within combustor are based on fully developed straight pipe channel estimation of heat load and flow characteristics, also known as Dittus-Boettler Equation:

\[
Nu_o = 0.023 \Re_{D}^{0.8} Pr^{0.3}
\]

However by comparing the fully developed straight pipe estimation values to the experimental results obtained in this project as plotted in Figure 4.15, the local heat load distribution showed a significant difference in values which is about 75-125% higher than the estimations.

Figure 4.15: Comparison of Local Nusselt Number Distribution to FDF Straight-pipe Estimation for Re=500000 and 662000

In addition, the lower Reynolds number (500k) case shows about 50% of underestimation from the fully developed straight pipe estimation than the higher Reynolds
number (600k) case does. However, it is also interesting to note that the results at the
downstream section of the combustor after the x=2D location merged to the same value for
both cases. It was believed that if the combustor length was long enough, the Nusselt
number values after the 2D location for both Reynolds number cases would meet at the
same value, which is around 40% higher than the fully developed straight pipe channel
estimation. These results led us in confidence to believe that the Nusselt number
distribution at any Reynolds number would have the same behavior and approach to the
same value as predicted. The results have provided a good agreement to Fully Developed
Flow phenomenon.

4.3 Comparison of Heat Transfer and Flow Results

Up to this point, we have proved that the peak locations for heat transfer and
turbulence intensity occurred at the same location respectively regardless of the Reynolds
number. It would be interesting to see if the heat transfer and turbulence intensity peak
locations share any relationship between them. As shown in Figure 4.16, by overlapping
the heat transfer coefficient distribution and the very-near-wall turbulence intensity
distribution together, the peaks have come to a great agreement with each other sharing the
same exact location. It clearly explains the close relationship between the heat transfer
coefficient and turbulence intensity.
Figure 4.16: The overlapping of Heat Transfer and Turbulence Intensity Distribution for $Re=500,000$ (top) and $Re=662,000$ (bottom)
CHAPTER 5: CONCLUSIONS

The heat transfer distribution and flow characteristics inside a “Dry Low NOx Emission” combustor equipped with a swirler, provided by Solar Turbines, Inc., were studied in an effort to understand the fundamental basis of flow behaviors that relate to heat loads within a turbine combustor to help improve the current cooling system, and design a more effective cooling system. Modern DLE combustors use minimal to zero film cooling air, and mostly back-side wall liner cooling to prevent localized hot and cold spots within the combustor causing higher NOx, CO and UHF emissions, thus the localized maximum heat load on the liner wall is the key factor to designing a proper cooling system using a focused back-side cooling technique. The goals are to prove the existence of localized heat load and to locate the concentrated heat load region at different Reynolds number conditions.

Both heat transfer and flow experimental results point to the same verdict that shows the existence of indisputable peak location of heat load and turbulence intensity. The results not only prove the existence of these peak locations, but also illustrate the locations of these peaks occur at the same spot from different experimental methodology and at different flow rates and Reynolds numbers. The heat transfer and flow characteristic results can precisely prove and support each other to further compliment the findings. It was also interesting to find out that the peak location does not vary with the Reynolds number. The peak location for heat load and turbulence intensity, as well as integral length- and time-scale, all appeared at the same spot regardless of the flow rate and Reynolds number. This leads us the conclusion that a localized focused cooling system can
be designed comfortably without having to worry about the variation in location with varying Reynolds number at real engine operation conditions.

However, the Reynolds number does have certain effects on both heat transfer and flow experimental results. It appeared that the swirling flow effect with wash-away phenomenon for higher Reynolds number (Re=662k) is relatively more significant than lower Reynolds number case (Re=500k) due to the stronger swirling air momentum created by stronger air flow at higher Reynolds number case. With the lack of strong swirling air momentum carrying enough energy, the air flow at lower Reynolds number (Re=500k) simply could not sustain the swirl throughout the combustor causing the flow to straighten out itself and die out so quickly before reaching through the end of combustor.

The phenomenon of the swirling effect causing a significant impact and difference on heat transfer and flow characteristics is evidently proven. It is experimentally shown from the results that the swirling flow possesses higher heat transfer capability than the axial flow does. If the level of heat transfer rate is rated in terms of flow directions, the tangential swirling flow will have the lead followed by the radial expanding flow then the parallel axial flow.

The explanations can easily be understood that the tangential swirling flow contains the centrifugal force spinning around within the combustor. The centrifugal force has the tendency to push the flow outward against the combustor wall; therefore it has a “sticky” effect to help the air flow “scrape” the heat off the combustor wall surface.

The heat transfer capability of radial expanding flow is believed to be higher than the axial flow as the expanding flow actually impinges directly on the combustor wall surface, which generally results in a higher heat transfer rate. However, due to the fact that
the expanding flow impingement only occurs at a small region at the upstream of the combustor, therefore the effect on an overall heat transfer is limited.

On the other hand, axial flow is the least capable in heat transfer because it travels parallel to the axis of the combustor, and causes the flow at a certain point to separate from the wall’s surface. Therefore the axial air flow has minimal contact with the combustor wall’s surface resulting in minimal heat transfer.

The flow measurement results also provide the evidence that the heat transfer peak location on the combustor liner effectively depends on the peak location of turbulent intensity of swirling flow and not the maximum value of the total velocity. The overall results have showed great agreements as predictions, and provided a better understanding of the relationship between heat transfer and flow characteristics.
REFERENCES


Yap-Sheng Goh was born in Pulau Pinang, Malaysia on May 1, 1979. In 1997, he graduated from Jit Sin (Independent) High School located in Bukit Mertajam, Pulau Pinang, Malaysia. Yap-Sheng then enrolled at Inti College Malaysia in the American University Twinning Program and was transferred to Louisiana State University in January of 2000 where he earned his Bachelor of Science degree in Mechanical Engineering in May of 2003. He continued his education at Louisiana State University to pursue a Master of Science degree in Mechanical Engineering under the guidance of Dr. Srinath Ekkad. Having fulfilled the requirements set forth by the Graduate School and the Department of Mechanical Engineering, Yap-Sheng will graduate in December of 2006.